

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

Muhammad Shahzad Masood, Atique Ahmad, Rizwan Alim Mufti

Mechanical Engineering Department, Pakistan Institute of Engineering and Applied Sciences Islamabad (PIEAS), Pakistan

Abstract— Biomechanics deals with the application of mechanics laws to living organism. Conventionally CAD models of complex biological shapes are prepared through Medical imaging software while in this paper an unconventional approach is employed for modeling of complicated geometry of human femur bone to make realistic investigations. Two orthogonal views are employed for modeling of complex femur bone with the help of 3D animated open source software Blender 2.63a. Pro/ENGINEER translated Blender prepared polygonal model into CAD model that is imported to ANSYS for analysis under different boundary condition. Results of stress analysis for heterogeneous bone structure vary with individuals that are beneficial for orthopedic surgeons.

Keywords— ANSYS; Biomechanics; Finite element analysis; Femur bone

1 INTRODUCTION

Biomechanics involves the application of mechanical principals to biological objects. It is not so simple to apply mechanical laws to biological objects. Artificial objects are simples in shape and they can be easily modeled whereas biological objects posses complex shape which are difficult to prepare CAD model. Nonlinear and anisotropic natures of biological objects are the source of difficulties in meshing and analysis [1].

Importance of computer simulations can't be denied in the field of biomechanics. Advanced processing power of computers has made it easy to simulate the biological objects possessing complexity in shape, nonlinearity and anisotropy in properties. Finite element method is the best way for linear, nonlinear and couple field analysis of biological objects. Traditionally MRI, CT scan and advanced imaging techniques were employed to prepare model for simulation purposes which is costly [2].

Femur bone is essential part of human body which provides support to human body while standing, sitting and walking. Cortical, compact tissues and other small parts is main constituent of Femur bone. Femur bone is complex in shape and it has different composition. The thickness of femur bone is in between 4-8mm and length is between 260-293mm [3]. Research has been carried out on femur bone fracture using mathematical modeling and different software. Somkid et al. performed modeling using CT scan and analysis was made using FEMLEB software [4]. Daan Waanders et al. studied fatigue creep damage at cement bone interface using finite element. T. San Antonio et al. focused on orientation of orthotropic materials properties in finite element model [5].

This study presents a new way of modeling the complex biological objects. Femur bone polygonal model was prepared with the help of Blender 2.63a. Reverse engineering feature of Pro/Engineer was beneficial for conversion of polygonal model into CAD model. This CAD model was imported to ANSYS 10 for analysis under different loading conditions.

sentation is a cumbersome task. Their complexity poses a great challenge in modeling. It is extremely difficult to prepare bone model using conventional CAD software. In this paper, Complex shaped femur bone model was prepared by employing an open source 3D animated software Blender 2.63a. Blender makes a polygonal model by approximating the surface of model with the polygons. Polygonal model is in the form of polygons mesh.

Extrusion modeling was adopted to make polygonal model of femur bone with the help of orthogonal views (front and side view). In extrusion modeling contours of object make 2D shape with help of one view. Then using 2nd view at some angle, a 3D shape is created. Femur bone model was prepared with quad face polygons and completed by scaling, rotating and extruding to collate the model with images.

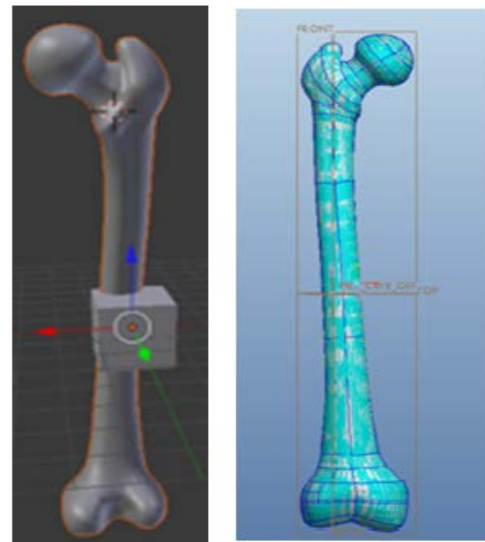


Figure 1 Blender Model (Left) and Pro/ENGINEER Model (Right)

2 FEMUR BONE MODELING

Due to the complex shape of bones their geometrical repre-

Blender prepared model consist of polygonal surfaces and CAD model possess NURB surfaces but CAD model is

necessary for finite element analysis of femur bone in the software ANSYS that's why Polygonal model of femur bone was converted into CAD model with the help of Pro/ENGINEER under restyle feature environment. Restyle is reverse engineering tool that has ability to create CAD model NURB surfaces on the top of polygonal surfaces of Blender model. Model can be imported into Pro/ENGINEER in different formats i.e. .dxf, .stl, .prt, wavefront etc. The model was exported in .stl format from Blender software and was imported to Pro/ENGINEER under wavefront format.

Restyle feature prepares the CAD model without losing any detail. NURB Surfaces on Polygonal model can be made using auto and manual patches option. But auto patches formation creates problems while meshing and analyzing results. So bone model was prepared using with manual surface. The model prepared is shown in Figure 1. The model created in Pro/ENGINEER was imported to ANSYS in .iges format. ANSYS 10 defeaturing option was utilized while importing the model.

3 MESHING AND SIMULATION

After importing the femur bone model to ANSYS, shell 181 element was selected for meshing because this element provide good results for linear, non linear, surface and solid model. Thickness was provided as real constant [6]. From the literature thickness of bone is described in the range of 4-8mm. In this research 5mm thickness was used.

After selection of shell 181 element in triangular form,

meshing operation was performed and boundary condition was applied. In real life bone can be broken under different loading condition. But in this paper three different boundary condition were applied.

- Applying pressure from upper side while restraint the bottom of femur bone
- Applying the pressure on bottom side while restraint the upper end of femur bone
- Applying the pressure on centre of bone while restraint upper and lower end of femur bone

In this paper the average weight of a person 72kg was used. The following material properties were used for linear static analysis [4].

$$\rho = 2000 \text{ kg/m}^3 \quad E = 2.130 \text{ E}9 \text{ Pa} \quad \nu = 0.3$$

4 RESULTS AND DISCUSSION

In the first case, pressure was applied on the upper side of femur bone while epicondyle was restrained (displacement boundary condition applied) as shown in Figure 2.

After performing simulation, deformations and Von Mises stresses were observed. The deformation and stress contour are shown in Figure 3. The maximum deformations occur on the upper side of femur bone and minimum deformation occurs on lower side of femur bone. The magnitudes of nodal deformations were 0.052604m. The maximum stresses were on the lower end of femur bone. The maximum stresses using Von Misses criteria were 5.7688E4 Pa.

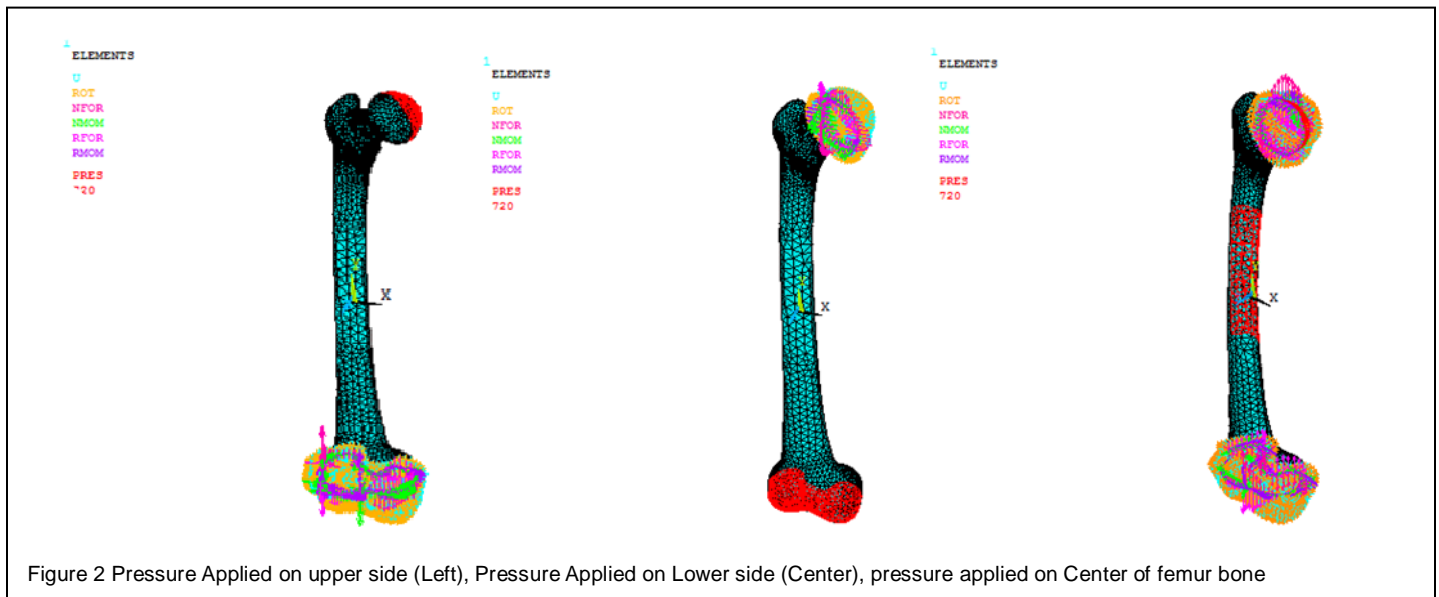
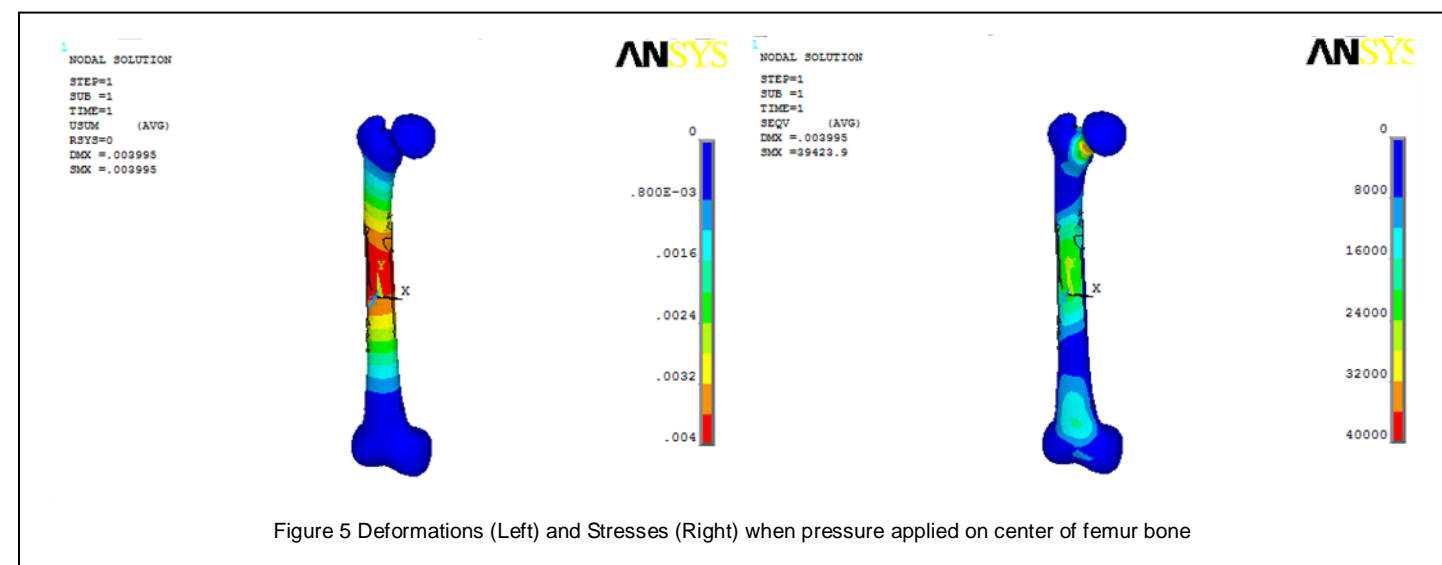
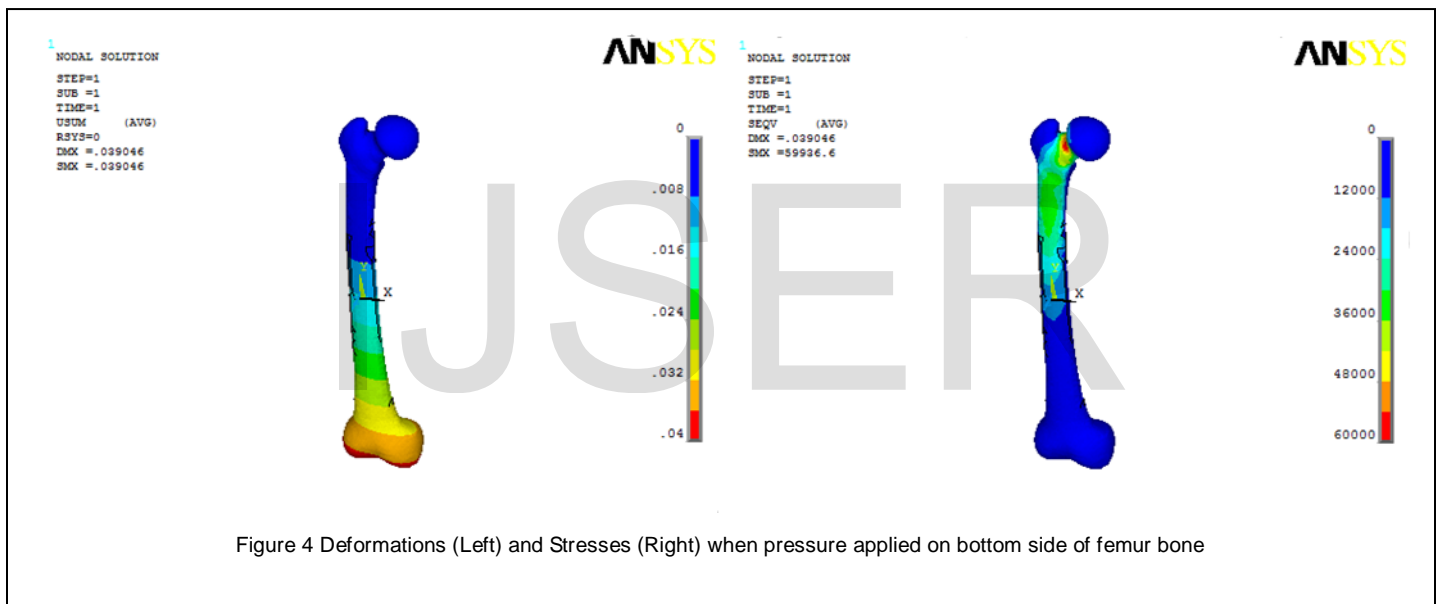
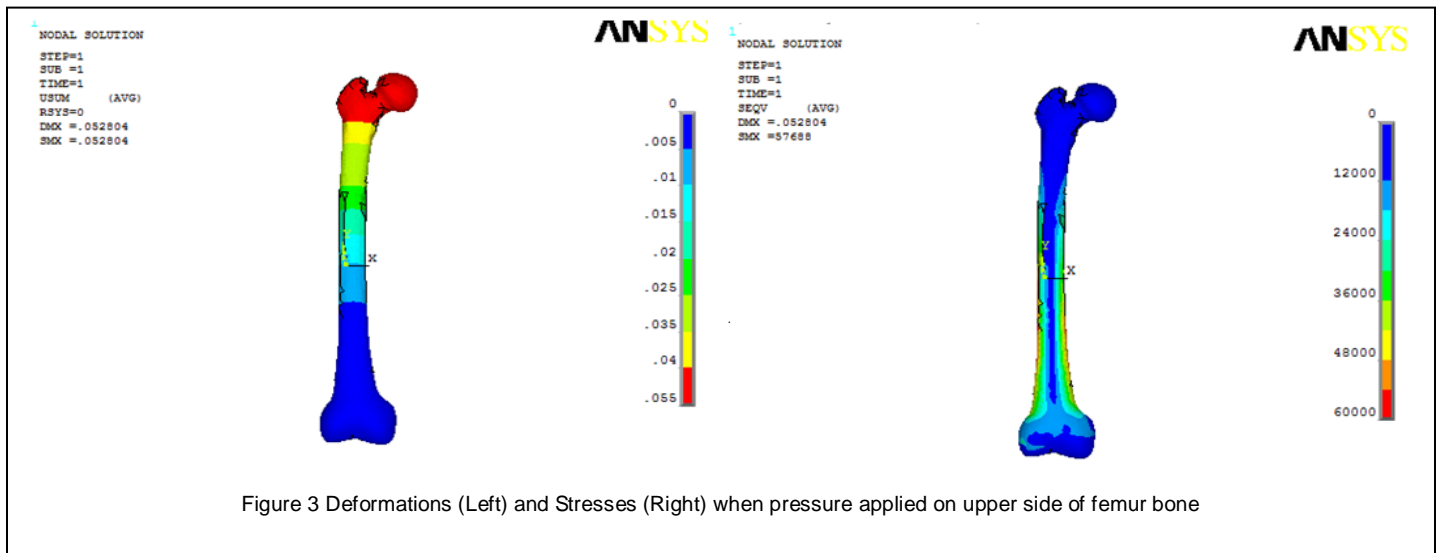


Figure 2 Pressure Applied on upper side (Left), Pressure Applied on Lower side (Center), pressure applied on Center of femur bone



In the second case, upper part of femur bone was restrained and pressure was applied on the epicondyle as shown in Figure 2.

The stresses were observed using von misses criteria. The maximum stresses were on the neck of femur bone while minimum stresses were on the lower part of the femur bone. The magnitude of maximum stress was 66596Pa. The maximum deformations were observed on the epicondyle and minimum on the upper part of the femur bone. The magnitude of maximum deformation was 0.039046m. Stresses and deformations were observed using nodal solution as shown in Figure 4.

In the third case, simulation was performed by restraining the bone from bottom and top. The pressure was applied in the center of bone as shown in Figure 2.

The stresses were observed using von misses criteria. The maximum stresses were on the neck of femur bone while minimum stresses were on the bottom of the bone. The magnitude of maximum stress was 59936Pa. The maximum deformation was found in the center while minimum deformation was on the epicondyle and on the head of femur bone. The magnitude of maximum deformation was 0.003995m. Stress and deformation contours are shown in Figure 5.

In all three cases it has been observed that maximum deformations are at that point where pressure is applied and deformations decreases as we go away from the area of application of pressure. Maximum stresses are located at restraint end and decreases towards free end. In the first and second case where pressure is applied at one end while restraint other end, bone is behaving as cantilever beam. In the case where pressure is applied in the centre of femur bone, it has been observed that magnitude of maximum deformation is 10 times lower than the other two cases.

CONCLUSION

In this paper a unique way of modeling and mechanical stress analysis of femur bone is focused. Modeling of Complex shape of femur bone is accomplished through the novel approach using a 3D animated software Blender. Pro/ENGINEER is beneficial while transforming polygonal model into CAD model. Simulation was performed after importing CAD model of human femur bone into ANSYS. Stress and deformation distribution varies with boundary condition. It has observed that Maximum stresses and minimum deformations are located at restraint end femur bone which indicates cantilever beam behavior of femur bone. Results vary with the individual that are helpful for orthopedic surgeons.

REFERENCES

- [1] D. Knudson, Fundamental of Biomechanics, springer, 2007
- [2] L. Voo, M. Armand, and M.kleinberger, Stress Fracture Risk Analysis of the Human Femur Based on Computational Biomechanics, Johns Hopkins APL Technical Digest, Vol. 25, No. 3, pp. 223, 2004
- [3] A. D. Sylvester, B. C. Merkl, and M. R. Mahfouz, Assessing A.L. 288-1 femur length using computer-aided three-dimensional reconstruction, Journal of Human Evolution 55, pp.665-671, 2008
- [4] S. Amornsamankul, K. Kaorapong, and B. Wiwatanapataphee,

Three-Dimensional Simulation of Femur Bone and Implant in Femoral Canal using Finite Element method, International journal of mathematics and computers in simulation. Vol. 4, No. 4, 2010

- [5] T. S. Antonio, M. Ciaccia and C. M. Karger, E. Casanova, Orientation of orthotropic material properties in a femur FE model: A method based on the principal stresses directions, Medical Engineering & Physics, Elsevier, 2011
- [6] ANSYS 10 Help