

# Biomechanical research of the femur finite element model combined with different material assignment methods

Wei Sheng<sup>1</sup>, Aimin Ji<sup>1,a</sup> and Changsheng Chen<sup>1,2</sup>

<sup>1</sup>College of Mechanical & Electrical Engineering, Hohai University, 213022 Changzhou, Jiangsu, China

<sup>2</sup>Changzhou Orthmed Medical Instrument Co., Ltd., 213022 Changzhou, Jiangsu, China

**Abstract.** The effects of different material assignment methods on femur biomechanical finite element analysis were explored, which provided guidance for material assignment in the study. According to different material assignment methods, the study was divided into the control group and experimental groups, the femur of the control group was assigned to 180 kinds of materials depend on the greyvalue, and the material assignment method of experimental group including the femur that assigned to 120, 60 and 10 kinds of materials depend on the greyvalue, the simplified femur model that assigned to two materials and the femur model that assigned to one material. Six groups different femoral finite element models was axial loaded and solved, and attain maximum equivalent stress of each group. The results showed that the experimental results of the material assignment schemes in the experimental group that including the femur that assigned to 120, 60 and 10 kinds of materials, and the simplified femur that assigned to two kinds of materials were consistent with the control group, while the experimental results of the femur that assigned to one material has obvious different with the control group. The analysis results show that when carries on femoral biomechanics finite element analysis, the femur evenly assigned to 10 kinds of materials depend on the greyvalue or simplified femoral model that assigned to two kinds of materials can be used, which can ensure the effectiveness and simplicity of the femoral material assignment.

## 1 Introduction

With the development of computer technology, finite element analysis of orthopedic biomechanics is becoming more and more popular. Determining the material properties is the prerequisite of finite element analysis of bones, and the prevailing approach is to assigned one uniform material. Zhang Junwen, etc. used uniform material assignment to study the effects that the size of the axial load on the biomechanical properties of the femur [1]. Wu Zehai, etc. also used uniform material assignment to research finite element analysis of the allogeneic bone plate combined with different screws in the treatment of femoral shaft fracture [2]. Because previous studies showed that there is a linear relationship between the CT image greyvalue and the bone density  $\rho$  [3], bone density and femoral unit material properties in empirical formula [4], so it can be regarded as a continuum, and the material assignment can be made according to the mutual relationship. Li Peng, etc. assigned 10 different material properties to the mandible depend on the CT value, efficient and accurate

---

<sup>a</sup> Corresponding author : wei\_321\_yue@163.com

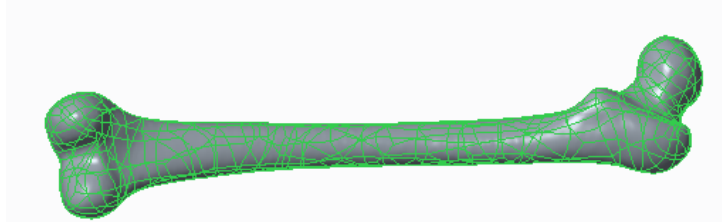
established the human mandible three-dimensional finite element model [5]. Fei Wanghua took the fresh pig femoral stem as the research object, established finite element model of fresh femoral stem based on CT images, which assigned to 802 kinds of different materials [6].

This study mainly start form the material assignment method depend on the greyvalue, the femur were assigned to 180 kinds of different material properties in the control group, and the biomechanical properties of the femur could be reduced by the assignment method of the control group. Regarding the control group as a reference, to explore whether it is can well reflect the biomechanical properties of the femur, when the femur were assigned to 120, 60 and 10 kinds of materials. At the same time, the material assignment method depend on the greyvalue must be conducted material assignment for body grid. When researching the femoral fracture internal fixation system, grid is relatively complex, femoral assignment is easy to get wrong. Therefore, this paper proposes a simplified femur model that assigned to two kinds of materials.

## 2 The establishment of the geometric model

Femoral data were obtained from a Philips / Brilliance 64-slice spiral CT scanner and the data were stored in DICOM format. DICOM standard format has been developed into an international standard for medical imaging information technology, which not only retains the real information of human femur, but also can conveniently communicate and exchange with different medical image devices. The DICOM data is directly imported into the MIMICS software, to determine the orientation of each image and obtain four engineering views. The threshold of the femur is set in the range of 226-2433, which distinguishes bones from other tissues. After the image editing, regional growing and 3D calculation, the femur model was obtained.

The femur model was imported into the Geomagic Studio to form the point cloud data. After the point processing, the polygon processing and the shape processing, the ideal curved surface model was formed. The NURBS surface model was transformed into a CAD model to complete the reconstruction of the femur. The femur geometry model can be directly applied to the analysis that the material assignment of the femur based on the greyvalue and femoral uniform assignment directly, femur geometry model is shown in figure 1.



**Figure 1.** The femur geometry model

By researching the characteristics of femur tissue, bone marrow was found in the central part of the femoral shaft. A 10 mm diameter hollow cylinder was set up in the femur model to simulate the bone marrow similar to the elasticity of water. And as the femoral cortical bone part of the very thin, a cancellous bone can be simulated by setting a ring cylinder with a thickness of 10 mm on the outside of the hollow cylinder, the rest are regard as cortical bone, the simplified femur model can be assigned to two materials in the subsequent processing. In summary, six group experimental geometrical models were established, and six groups femur models were introduced as shown in Table 1.

**Table 1.** Introduction of the femur models of different group

Group	Introduction
-------	--------------

Group 1 (control)	The femur was evenly assigned to 180 materials by the greyvalue
Group 2 (experimental)	The femur was evenly assigned to 120 materials by the greyvalue
Group 3 (experimental)	The femur was evenly assigned to 60 materials by the greyvalue
Group 4 (experimental)	The femur was evenly assigned to 10 materials by the greyvalue
Group 5 (experimental)	The simplified femoral model was assigned to two materials
Group 6 (experimental)	The femur was uniformly assigned to one material

### 3 The establishment and solving of the finite element model

#### 3.1 Meshing and material assignment

When the finite element model of material assignment depend on greyvalue is established, the surface grid meshing of the three-dimensional model of the femur is completed by MIMICS. The parameters are set according to the default value, and the surface grid is saved as Ansys element format. Then, the surface grid file is imported into Ansys, and complete volume grid meshing based on surface grid. The nodes, elements and PREP7 files were written based on the volume grid respectively. In the FEA module of MIMICS, read nodes, elements and PREP7 files, select solid 92 as the element type, and use material function to assign the material value depend on greyvalue of the CT image, the concrete expression as shown in follow equations[7]:

$$Den = -13.4 + 1017 * Gra \quad (1)$$

$$E-Mod = 388.8 + 5925 * Den \quad (2)$$

$$Poi = 0.3 \quad (3)$$

Where, *Den*, *Gra*, *E-Mod* and *Poi* are the density, grey value, e-modulus and poisson's ratio of the femur.

When the finite element model was set up in group 5 and 6, femur and simplified femur model were respectively imported into the finite element analysis software Ansys to set the element type and material property directly, and to complete the division of the volume grid meshing. the element type and Material properties are set as shown in Table 2.

**Table 2.** The setting of element type and material properties [2,8]

Group	Elastic Modulus [MPa]		Poisson's ratio	Element type
Group5 (experimental)	Cortical bone	Cancellous bone	0.3	Solid 92
	15000	1100		
Group6 (experimental)	12000		0.3	Solid 92

### 3.2 Loading with solving

A loading node was created above the femoral head, and a rigid region was established between the node and nodes included in the femoral head surface. Considering the axial force of the femur under normal standing and walking, the axial load of 600N and 1000N was applied to the loading node, and the displacement of the distal face end of the femur was restrained. The default solver is used to calculate the finite element model, and the corresponding equivalent stress contour map is obtained.

### 4 Results

According to the finite element analysis model established, the biomechanical analysis of 12 groups experiments was completed. The results of the experiment are shown in Figure 2. The maximum equivalent stress in the control group was 13.4431Mpa, when standing by one leg, the group 6 has a difference with the control group, but the other groups are close to the control group, the maximum equivalent stress contour map in the control group is shown in Figure 3. In the walking condition, the maximum equivalent stress of the control group was 21.845MPa, and the performance of the experimental group was consistent with standing by one leg, the maximum equivalent stress of the control group was shown in Figure 4.

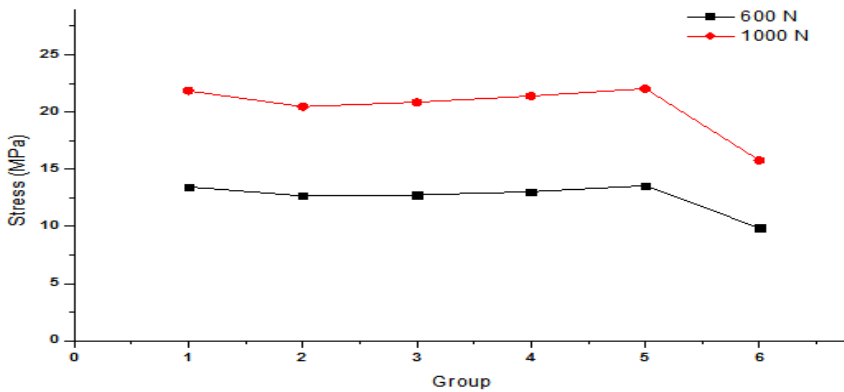


Figure 2. The maximum equivalent stress of each group femur under the different axial loading

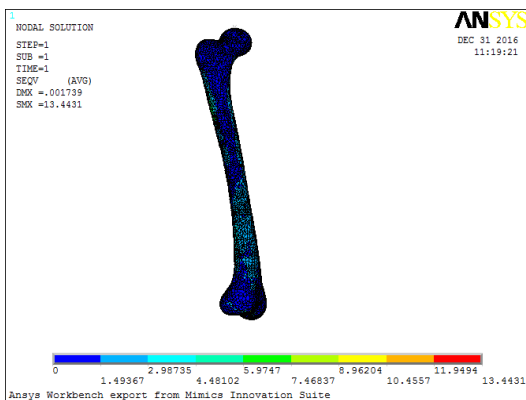


Figure 3. Equivalent stress contour map of the femur in the control group under the 600N axial loading

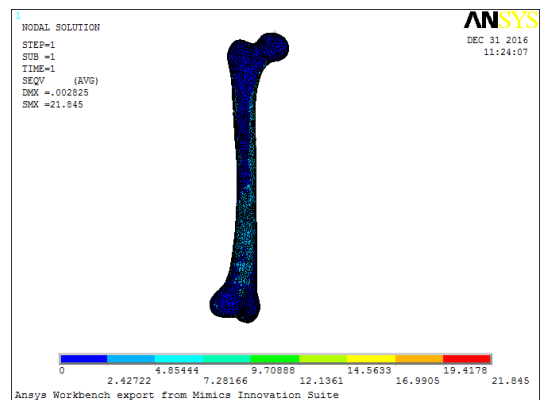


Figure 4. Equivalent stress contour map of the femur in the control group under the 1000N axial loading

## 5 Discussion

In this study, finite element method was used to evaluate femur biomechanics of different material assignment. Finite element method was applied to the study of orthopedic biomechanics since 1972, and has made great achievements in the orthopedic and orthopedic medical devices biomechanics research. At the same time, the finite element method has the advantages of cost-saving, time-saving, high-efficiency, high simulation and repeatability, which has an irreplaceable role in other biomechanical experimental methods, and be used in bone biomechanics research increasingly to provide guidance for clinical research[9].

When using finite element method to conduct the orthopedic biomechanical analysis, the bone material assignment is one of the most important step. The goal of femur material assignment is not only to guarantee the accuracy of model material assignment, but also to realize the simplicity and economic of assignment operation. This research is aimed at this goal, through the biomechanical finite element analysis for the femur that using different material assignment methods, to choose the most simplicity and economic material assignment method based on the validity of the assignment.

The analysis of the research results show that the biomechanical effects of the femur evenly assigned to 120, 60, or 10 kinds of materials depend on the greyvalue in the experimental group were consistent with the femur assigned to 180 kinds of materials in the control group, no matter how much the axial load is. So if you want to assign femur's material depend on the greyvalue, the femur could be assigned to 10 kinds of materials directly. When the simplified femur model was assigned to two kinds of materials in the experimental group, it was consistent with the biomechanical effect of the control group. But when the femur model was assigned to one unified material in the experimental group, it has a quite different biomechanical effect with the control group. When there is no greyvalue data or hoping to find a more convenient material assignment method, the simplified femoral model that assigned to two materials can be used to ensure the biomechanical properties of the femur.

In this study, under the premise of guaranteeing the accuracy of the research results, the finite element analysis model is simplified boldly, which can ensure the simplicity and generality of the research. The only major axial force in the external loading was considered, but the actual situation of fracture stress is often affected by muscle and ligament and other soft tissue effects [10]. Although there is little need to be improved in the whole finite element analysis process, but the analysis results has a quite consistent with the real situation. In the following study of this topic, we will gradually improve the shortcomings of the study, and strive to provide a more convenient and effective research methods.

## 6 Conclusions

Based on the finite element analysis of biomechanics, the following conclusions can be drawn from the research and discussion:

- (1) The finite element analysis (FEA) method has been successfully applied in this study, the data obtained were matched to the relevant human femoral experiments, and the validity of the results of finite element analysis can also ensure the validity of the conclusion of the study.
- (2) When conducting finite element analysis of the femur and needing to assign material to the femur depend on the greyvalue, to evenly assign 10 kinds of materials to the femur is a good choice, which can ensure the simplicity and the validity of the material assignment at the same time.
- (3) When the researchers were unable to obtain the CT data of the femur or were unable to perform the operation of the MIMICS software, the study shows that the use of simplified femoral model and the assignment method can also be effective in the material assignment of femur and research of corresponding finite element analysis.

## Acknowledgements

This work is financially supported by the central university basic scientific research business expenses special funds (No. 2016B49614), the innovation training project of graduate student in jiangsu province (No. KYLX16\_0708) and the special fund project of the transformation of scientific and technological achievements in jiangsu province (No. BA2012061).

## References

1. Zhang Junwen, Deng Shuangcheng, Zheng Haiyang, et al. The Establishment and Static Analysis of the Finite Element Model of the Human Femur[J]. Journal of Beijing Institute of Petrochemical Technology, **20(3)**, 45-47(2012).
2. Wu Zehai, Wang Yongqing, Dong Limin, et al. Finite Element Analysis of Allogeneic Bone Plates Combined with Different Screw in the Treatment of Femoral Shaft Fractures[J]. Journal of Practical Orthopaedics, **20(10)**, 910-913(2014).
3. Rho JY, Hobatho MC, Ashman RB. Relations of Mechanical Properties to Density and CT Numbers in Human Bone[J]. Med Eng Phys, **17(5)**, 347-355(1995).
4. Wirtz DC, Schiffers N, Pandorf T, et al. Critical Evaluation of Known Bone Material Properties to Realize Anisotropic FE-simulation of the Proximal Femur[J]. J Biomech, **33(10)**, 1325-1330 (2000).
5. Li Peng, Liang Rui, Shen Longduo, et al. Establishment of Three-dimension Finite Element Model of Human Mandible with MIMICS and ANSYS Software[J]. J Pract Stomatol, **28(6)**, 709-713(2012).
6. Fei Wanghua. Finite Element Analysis and Experimental Verification of Fresh Bone CT Images[D]. Nanjing University of Science and Technology, (2008).
7. Zhang Guodong, Liao Weijing, Tao Shengxiang, et al. Methods for Material Assignment of Finite Element Analysis with Femurs[J]. Journal of Rehabilitative Tissue Engineering Research, **13(43)**, 8436-8441(2009).
8. Gao Yalei. Finite Element Analysis and Experimental Verification of Titanium Plate Fixation for Femoral Fracture System[D]. Tianjin University of Technology, (2014).
9. Zhang Yan, Yang Tieyi, Liu Yishu, et al. Stress Distribution of Proximal Screw Fixation of Single and Double Cortex Different Combinations in Less Invasive Stabilization System-distal Femur Treatment: A Finite Element Analysis[J]. Chinese Journal of Tissue Engineering Research, **16(4)**, 581-584(2012).
10. Zhou Xingheng, Wang Yongqing, Dong Limin, et al. Finite Element Analysis of Proximal Tibial Fractures Fixation with Interlocking Intramedullary Nail and Locking Compression Plate[J]. Journal of Practical Orthopaedics, **20(11)**, 995-998(2014).