

# The Effect of Material Inhomogeneous for Femoral Finite Element Analysis

Chih-Han Chang\* Jui-Ting Hsu Shou-I Chen Weng-Pin Chen<sup>1</sup> Guan-Liang Chang

*Institute of Biomedical Engineering, National Cheng Kung University, Tainan, Taiwan, 701, ROC*

<sup>1</sup>*Department of Biomedical Engineering, Chung Yuan Christian University, Chung-Li, Taiwan, 320, ROC*

Received 30 Aug 2001; Accepted 24 June 2002

---

## Abstract

The accuracy of a finite element simulation depends mostly on the mesh model. Many parameters influence the precision of a finite element mesh. In orthopedic biomechanics, the material property of bone is one of the major parameters in the simulation, especially the heterogeneous of cortex and cancellous bone. The common approach to simulate the heterogeneous of bone is to model with several sets of elastic modulus for the bone material. However, the required set numbers of elastic modulus to obtain a feasible solution is still unclear for bone structure. The objective of this study was therefore to investigate the necessary of heterogeneous femoral material for finite element simulation. Three rabbit femora were selected as the target subjects and three-point bending test was used as the loading mode. Experimental measurement of the displacement value under the three-point bending test was used as the reference point for justification the analytical results and convergence. The results indicated that the displacement value decreased as the set number of cortex elastic modulus reduced. To achieve a 5% absolute convergence for displacement, a minimum of 60 sets of cortex bone elastic modulus is required.

**Keywords:** Finite element method, Femur, Heterogeneous, Three-point bending test.

---

## Introduction

Since Breakelmans first introduced the finite element analysis for spinal simulation in 1972, the finite element method has become an important research tool in orthopedic biomechanics. Finite element method, comparing with the experimental approach, can provide comprehensive predication of the full field mechanical indices and allow effective control of the investigated parameters in the analysis thus is suitable for studying the biomechanical rational of orthopedic structures. With the advance of computer power, the applications of finite element simulation have been extended to the field of non-linearity such as the investigation of the bone fixation system [1], evaluation of fracture risk for osteoporosis [2,3] or even predication of bone remodeling after total hip replacement [4]. The efficiency of the finite element analysis depends on the finite element models. However, the process of setting up the finite element models in orthopedic applications still lacks a rigid guideline.

In early orthopedic applications, two-dimensional models were the major approaches due to the limitations in computer capacities and the relative ease in setting up the geometry

and loading conditions [5,6]. These simplified models are difficulty to provide valuable clinical information. Recently, three-dimensional approaches have become the major trend for orthopedic finite element simulation [3,7,8]. The typical procedures to setup a three-dimensional mesh model, especially for the bone structure, is to employ CT images. Nodes are placed on the boundaries of bone structure with hand-digitizer or simple interval dividing programs [8]. Corresponding nodes between CT slices are then connected to form three-dimensional elements. These processes not only require intensive labor efforts but also produce ill-conditioned elements. With the advent of computer graphics techniques and computer aided design systems, many algorithms were developed to generate the three-dimensional bone mesh with less human interaction and offer better element shape.

Even with the better approximation in bone geometry by these semi-automatic mesh generators, the properties of bone material still provide obstacles in setting up a realistic finite element bone mesh. The bone material has been recognized as nonlinear, anisotropy and non-homogeneity; but in most finite element models, the bone material is generally modeled as linear, isotropy and assigned with only two material moduli, that is cortex and cancellous [5,6]. The nonlinear and anisotropic characteristics of the bone material are still difficulty to be simulated in finite element models

---

\* Corresponding author: Chih-Han Chang

Tel: +886-6-2757575 ext. 63427; Fax: +886-6-2343270

E-mail: cchang@mail.bme.ncku.edu.tw

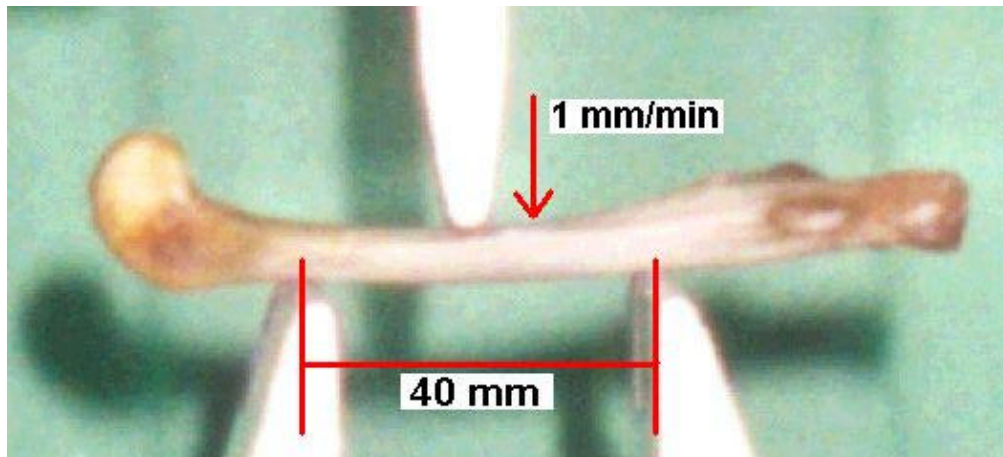


Figure 1. Schematic plot of the three-point bending test

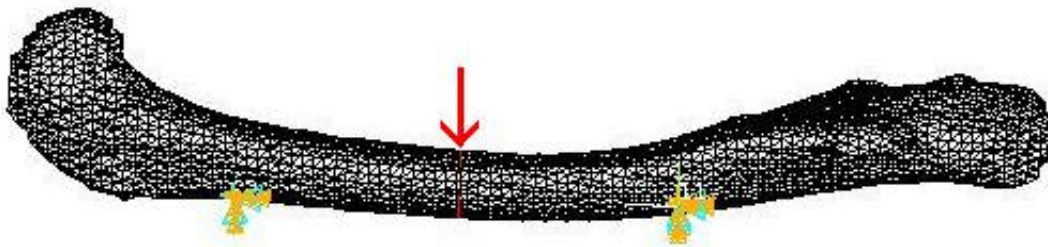


Figure 2. The finite element mesh model to simulate the three-point bending

at current stage. The heterogeneous property of bone is possible to be modeled with finite number of elastic modulus based on the CT data. The objective of this study was therefore to investigate the heterogeneous modeling of bone material for finite element analysis. To justify the simulated results, displacement measurements of rabbit femora using three-point bending test were employed as the absolute convergent reference point.

### Material and Methods

Three major phases were included in this study. Firstly, rabbit femora were used to execute the three-point bending experiments and the force-displacement curves were recorded to setup the reference for the validation of finite element simulations. After that, three-dimensional finite element models for the corresponding three-point bending tests were generated by a self-develop of program based on CT section images [9]. Bone material properties of these models were assigned by the power law relationship between elastic modulus and CT numbers [7]. Finally, various numbers of bone material, simulated with different elastic modulus, were applied in the finite element models to evaluate the effect of non-homogeneous bone material.

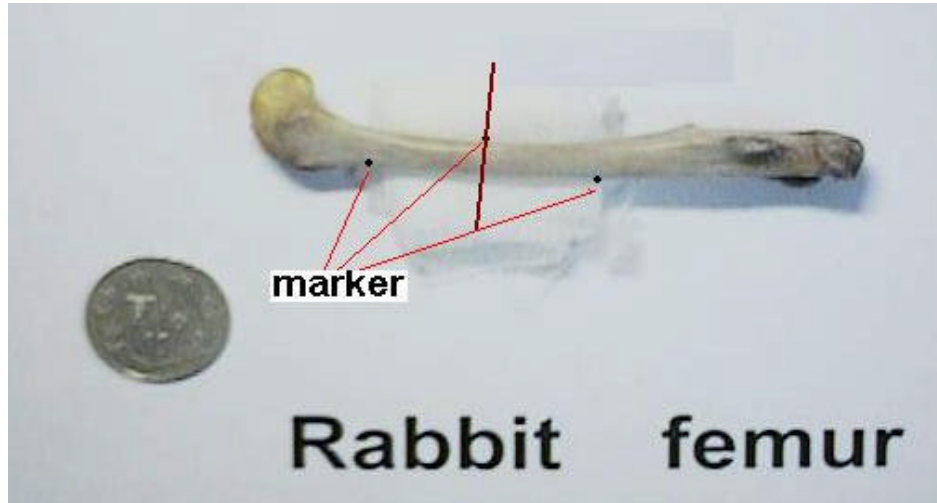
### Three-point Bending Test

Three rabbit femora were used in this study including two normal femora and an osteopropostic femur induced by immobilization. After sacrifice of the rabbits, the soft-tissue

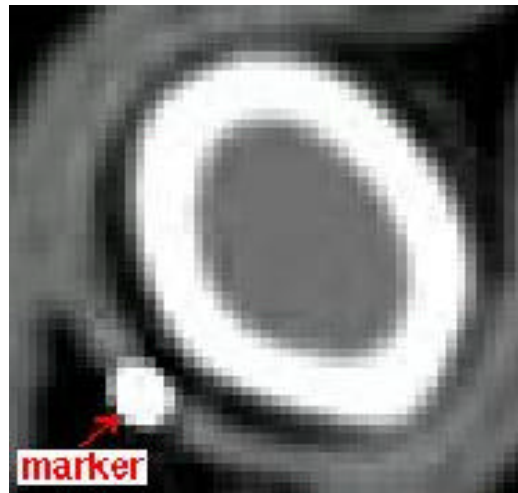
were stripped from the femora and scanned with GE fast CT (General Electric Co., USA) with 1mm slice interval and 512x512 resolution at 15mm of field of scan (FOS). In order to maintain the bone quality, the three-point bending test was performed within 1hr after each sacrificing. Before the test, each femur was warped by wet clothes and stored inside a sealed container. The experimental set up of the three-point bending test was shown in Figure 1. The loading speed was set at 1mm/min till fracture. The force-displacement curves of the three femora were recoded as the reference for finite element validation.

### Three-dimensional Finite Element Mesh

For the three femora, three dimensional mesh models were generated through a self-develop of program based on the CT images of each femur [11]. In order to have an accurate approximation in the femoral geometry, the average element size for the tetrahedron elements was less than 0.6 mm. Figure 2 shows one of the three mesh models. To better simulate the three-point bending test for finite element analysis, three markers were placed on each femur before CT scanning to identify the three loading locations during bending test (Figure 3). These markers detectable in the CT images were used in each mesh model to setup the loading and constrained boundary conditions in the three-point bending simulation. The nodes at the two lower positioned markers were fixed in all directions as the boundary condition; while the node matched the upper marker position was loaded by a concentration downward force of 136.3N is one step as show in figure 2.



(a)



(b)

Figure 3. (a) The rabbit femur with three markers to identify the locations for the three-point test. (b) The CT image of rabbit femur with the marker observed.

#### Assignment of modulus of Elasticity

Many studies have revealed that there is a power relationship between the elastic modulus and the apparent density of bone material, and a linear correlation between the apparent density and CT number [3,7,12]. In this study, the elastic modulus of each tetrahedron element was derived from the CT numbers at its four nodes based on the following equation:

$$E = A*(CT/100) B \quad (1)$$

where E: elastic modulus, CT: CT number and A, B: material parameters.

The elastic modulus of each node, with its CT number obtained from the CT images, was calculated from above equation and the elastic modulus of a tetrahedron element was acquired from the averaged elastic modulus of the four nodes that construct this element. The values of parameters A and B for each femur were obtained by a trail and error approach to

match the displacement measured from the three-point bending test and the finite element simulation on that particular femur.

To study the non-homogeneous effect of bone material in the finite element models, fourteen groups of various numbers of elastic modulus were setup as shown in Table 1. Within the table, X stands for the number of cortex bone materials and Y identifies the number of spongy bone materials used in each femoral model. The CT range of cortex bone was assigned between 1500 and 3000 and the CT range of cancellous bone was assigned within 800 to 1500. Therefore, in the first group of Table 1, each CT number has a corresponding elastic modulus and this is the most non-homogeneous model available within the resolution of CT. The last (fourteenth) group of Table 1 would only have one material for cortex and one for cancellous respectively and this represents the most homogeneous model. The Poisson's ratio for all materials was assigned to 0.3 as referred from literatures [8, 9]. All materials were assumed to be linear elastic.

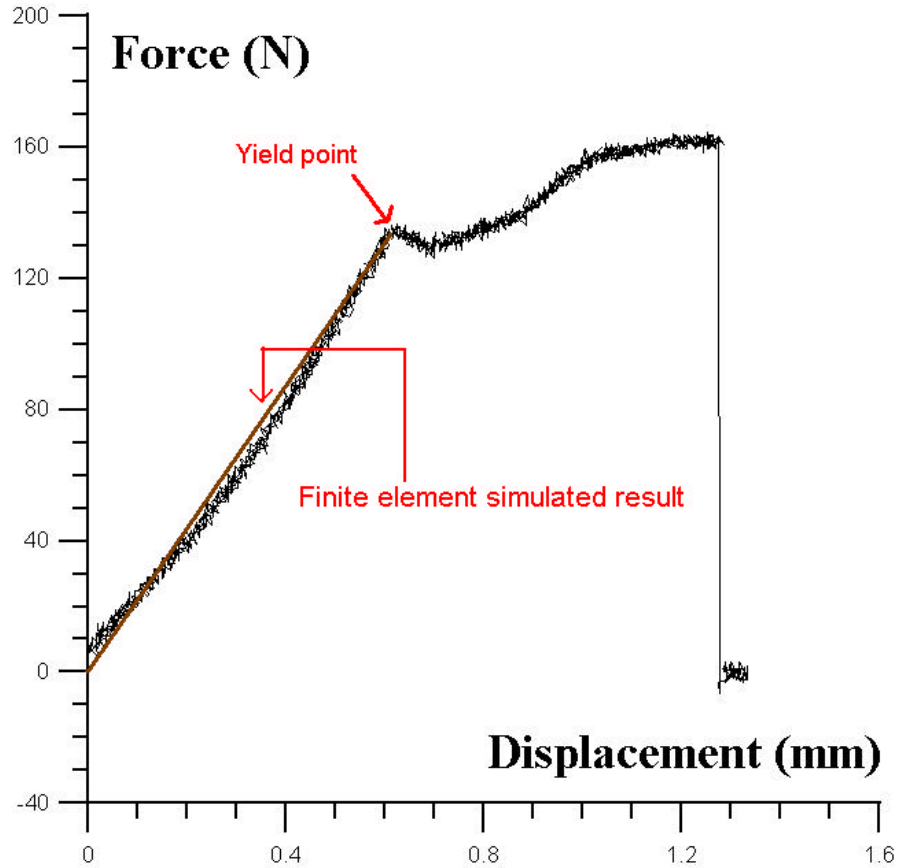


Figure 4. The force-displacement curve of a rabbit femur under three-point bending test and finite element simulated results

Table 1. Fourteen groups of different levels of heterogeneous femur models were setup. The first group represents the highest level of heterogeneous and the level of heterogeneous decreased as the group number increased.

Group	Cortical bone material numbers (X)	Cancellous bone material numbers (Y)	Displacement (mm)
1	2000	700	0.625
2	1000	350	0.624
3	500	175	0.622
4	250	88	0.618
5	200	70	0.616
6	125	44	0.611
7	63	22	0.597
8	32	11	0.571
9	20	7	0.545
10	16	6	0.525
11	8	4	0.455
12	4	2	0.359
13	2	1	0.253
14	1	1	0.156

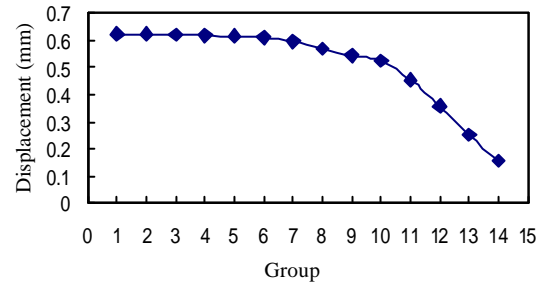
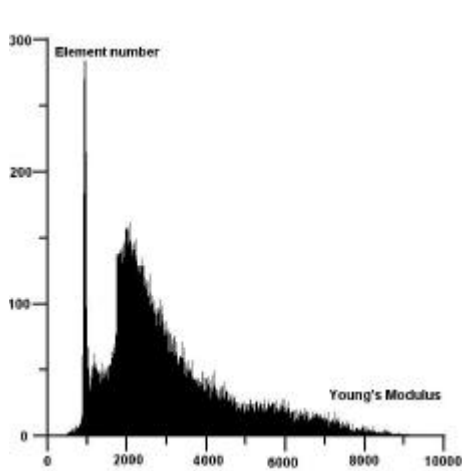


Figure 5. The displacement values decreased as the level of heterogeneous reduced

**Results**

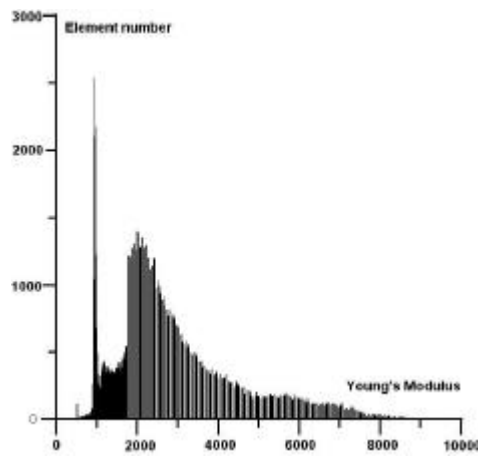
Figure 4 shows the force-displacement curve of a femur under three-point bending test and the fracture force of this femur was 162.5 N. The yield point was determined at the end of the linear portion of the curve and the values for yield force and displacement were 136.3 N and 0.624 mm, respectively. With these measured values, the parameters of A and B in the equation for elastic modulus could be determined for the group one (the most non-homogenous) model, i.e., with 2000 cortex materials and 700 cancellous materials. Two sets of A and B (A=7.74, B=2.0 and A=4.09, B=2.2) were obtained by the trail-





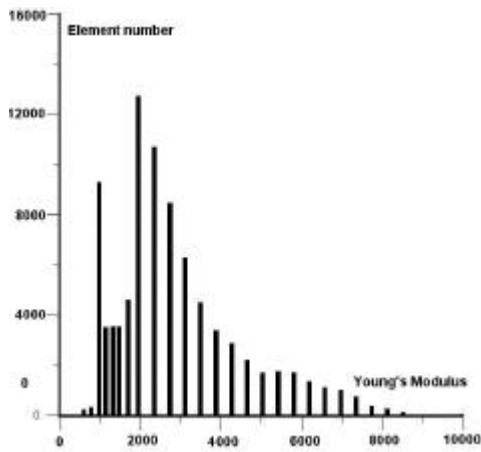
FEM displacement = 0.62383 mm  
 average cortical bone E = 3227 MPa  
 average cancellous bone E = 1063 MPa  
 average intact bone E = 2621 MPa

(a)



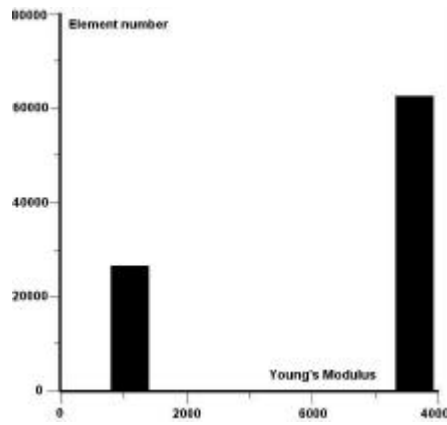
FEM displacement = 0.61443 mm  
 average cortical bone E = 3227 MPa  
 average cancellous bone E = 1063 MPa  
 average intact bone E = 2621 MPa

(b)



FEM displacement = 0.53595 mm  
 average cortical bone E = 3285 MPa  
 average cancellous bone E = 1061 MPa  
 average intact bone E = 2626 MPa

(c)



FEM displacement = 0.14344 mm  
 average cortical bone E = 5981 MPa  
 average cancellous bone E = 989 MPa  
 average intact bone E = 4438 MPa

(d)

Figure 6. The histogram of number of elements for each elastic modulus in (a)group 1 (b)group 5 (c)group 9 (d)group 14





and-error approach, which matched the simulated displacement with the experimental yield point.

For the first set of parameters,  $A=7.74$  and  $B=2.0$ , the displacement value equals  $0.625\text{mm}$  (under  $136.3\text{N}$  force) which indicates a acceptable simulated result as compared to the experimental yield point as show in figure 4. However, as the numbers of elastic modulus reduced in each material group simulation, the displacement values decreased (under the same  $136.3\text{N}$  bading) as shown in Table 1 and figure 5. It is observed that as the number of elastic modulus was less than 20 for the cortex, the displacement demonstrates a significant decreasing. To achieve a 5% convergence for displacement the number of cortex elastic modulus should greater than 60.

For the second set of parameters,  $A=4.09$  and  $B=2.2$ , and the rest two femora, the results were similar, i.e., as the number of elastic modulus reduced in the simulation, the displacement decreased (under the same loading) and the minimum number of cortex elastic modulus should greater than 60 in order to achieve a 5% convergence in displacement.

### Discussion and Conclusion

Three factors affect the accuracy of a finite element model, i.e., the geometry, the material and the loading (including boundary) conditions. In this study, with better approximation in geometry and loading condition, the heterogeneous of femur was evaluated to justify its influence.

When the elastic modulus is converted from CT number, which is the general approach adopted by most finite element researches, under the same loading force, the displacement decreased as the number of cortex elastic modulus reduced. To explain the results, the histograms of the elastic modulus distributions of four material groups were shown in Figure 6. For the group one model, the highest level of heterogeneous, the simulated displacement value was identical to that of the experiment and the averaged elastic modulus for the cancellous, cortex and the whole femur were  $1063\text{Mpa}$ ,  $3277\text{Mpa}$  and  $2621\text{Mpa}$ , respectively. For the model with 200 cortex materials and 70 cancellous materials, there was roughly 1% of error in the displacement outcome. However, for the model with only 20 cortex and 7 cancellous materials, the displacement error was greater than 10% while the difference in averaged elastic modulus (compared with group one) was less than 1% ( $1061\text{Mpa}$ ,  $3285\text{Mpa}$  and  $2626\text{Mpa}$  for cancellous, cortex and whole femur, respectively). For the most homogeneous model (group 14), the averaged elastic modulus ( $989\text{Mpa}$ ,  $5981\text{Mpa}$  and  $4483\text{Mpa}$  for cancellous, cortex and whole femur respectively) was much greater than the value in material group one hence induce a much smaller displacement result.

To assess the obtained elastic modulus, the analytical equation (as showed bellow) for three-point bending test was employed.

$$E = \frac{P \times L^3}{48 \times Y_{\max} \times I} \quad (2)$$

The moment of inertia was calculated from the average of three CT section images. On each image, the cortical cross-section was approximated by an ellipse. The values of averaged  $I$  ( $I = 74.3 \text{ mm}^4$ ), the loading ( $L = 136.3 \text{ N}$ ), the length ( $L = 40 \text{ mm}$ ), and the displacement ( $Y_{\max} = 0.624 \text{ mm}$ ) were taken into the above equation, and the corresponding elastic modulus was  $3920\text{Mpa}$ , which approximated the elastic modulus of cortex in material group one ( $E=3227\text{MPa}$ ).

From figure 6, it could also be observed that the influence of cortex is larger than the effect of cancellous because the change on average cancellous elastic modulus has less impact on the displacement result than the cortex. The reason is that under three-point bending test the cortex shell located at the outer of the femur would contribute more on the moment of inertia, also the loaded portion of this model is at the femoral mid-region, where few cancellous bone existed. To justify this reason, the number of elastic modulus of cancellous material in each material group was reduced by 10 times. The result showed that the differences in displacement values were less than 1%. From this observation, it is noted that the heterogeneous material effect of this study should work on the cortex bone only.

The results of this study indicated that the heterogeneous material distribution of femoral cortex plays a vital role in biomechanical researches, especially when the elastic modulus are converted from CT number in finite element simulations. A minimum selection of 60 cortex materials is required to simulate the heterogeneous material distribution of femur for a 5% displacement convergence, for the convergence of strain or stress, the heterogenous might be more critical.

### References

- [1] W. M. Mihalko, A. J. Beaudoin, J. A. Cardea and W. R. Krause, "Finite - element modeling of femoral shaft fracture fixation techniques post total hip arthroplasty", *J. Biomech.*, 25(5): 469-476, 1992.
- [2] J. C. Lotz, E. J. Cheal and W. C. Hayes, "Fracture prediction for the proximal femur using finite element models: part I - linear analysis", *J. Biomed. Eng.*, 113: 353-360, 1991.
- [3] J. H. Keyak, S. A. Rossi, K A. Jones and H. B. Skinner, "Prediction of femoral fracture load using automated finite element modeling", *J. Biomech.*, 31: 125-133, 1998.
- [4] H. Weinans, R. Huiskes, B. von Rietbergen, D. R. Sumner, T. M. Turner and J. O. Galante, "Adaptive bone remodeling around bonded noncemented total hip arthroplasty: a comparison between animal experiment and computer simulation", *J. Orthopaedic Research*, 11: 500-513, 1993.
- [5] H. Weinans, R. Huiskes and H. J. Crootenboer, "Trends of mechanical consequences and modeling of a fibrous membrane around femoral hip prostheses", *J. Biomech.*, 23: 991-1000, 1990.
- [6] P. J. Prendergast and D. Taylor, "Stress analysis of the proximo - medial femur after total hip replacement", *J. Biomed. Eng.*, 12: 379-382, 1990.
- [7] J. H. Keyak, J. M. Meagher, H. B. Skinner and C. D. Mote, "Automated three - dimensional finite element modeling of bone: a new method", *J. Biomed. Eng.*, 12: 389-397, 1990.
- [8] T. P. Harrign and W. H. Harris, "A finite element study of the effect of diametral interface gaps on the contact areas and pressures in uncemented cylindrical femoral total hip components", *J. Biomech.*, 24(1): 87-91, 1991.

- [9] J. D. Reubrn, C. H. Chang, J. E. Akin and D. R. Lorberger, "A knowledge-based computer-aided design and manufacturing system for total hip replacement", *Clin. Orthop.*, 285: 48-56, 1992.
- [10] K. Stromsoe, A. Hoiseth, A. Alho and W. L. Kok, "Bending strength of the femur in relation to non-invasive bone mineral assessment", *J. Biomech.*, 28(7): 857-861, 1995.
- [11] C. L. Lin, C. H. Chang, C. S. Cheng and H. E. Lee, "Three-dimensional finite element mesh generation for maxillary second premolar", *Comput. Methods Programs Biomed.*, 59: 187-195, 1999.
- [12] D. R. Carter and W. C. Hayes, "The compressive behavior of bone as a two-phase porous structure", *J. bone Joint Surg. Am.*, 59: 954-962, 1977.
- 



# 股骨非均質材料特性對有限元素分析之影響

張志涵\* 許瑞廷 陳守義 陳文斌<sup>1</sup> 張冠諒

國立成功大學醫學工程研究所

<sup>1</sup>中原大學醫學工程學系

收件日期 2001年 8月30日；接受日期 2002年6月24日

## 摘 要

有限元素模擬中，模型建立的好壞是影響模擬正確性與否的重要因素。影響模型好壞的因素很多，在骨科生物力學中，骨骼材料性質的設定是一重要因素。以往研究中有限元素模擬模型大多是假設骨骼為均質性或近似非均質性的材料，但這和骨骼非均質的特性有很大的不同，若以多組不同彈性係數來模擬骨骼的非均質性，則需要幾組才是恰當？當組數不同時其影響又為何？都有待探討。本研究以相同幾何外型但不同材料數目的股骨有限元素模型，以三點彎距（three-point bend）位移量的比對，探討股骨有限元素分析中皮質骨(cortex)材料非均質性的影響。模型中材料性質的判定，是利用電腦斷層掃描影像的亮度值和彈性係數（elastic modulus）關係轉換而得，由最多種材料數目開始做不同程度的減少，探討模型中材料數目對模擬結果影響的分析。並且以成兔股骨實驗量測three-point bending位移量，和對應的有限元素模型分析比較。結果顯示，模擬分析three-point bending之位移量會隨著模型材料數目減少而下降，比對實驗結果，建議在做骨骼有限元素模擬時，皮質骨的材料數目最少要超過60種，才可以得到位移量有5%收斂。

關鍵詞：有限元素法、股骨、非均質性、三點彎距

\* 通訊作者：張志涵

電話：+886-6-2757575 ext. 63427；傳真：+886-6-2343270

電子郵件信箱：cchang@mail.bme.ncku.edu.tw

