FEMORAL STEM SHAPE DESIGN OF ARTIFICIAL HIP JOINT USING A VOXEL BASED FINITE ELEMENT METHOD

Taiji ADACHI [*], **Hiromichi KUNIMOTO** [†], **Ken-ichi TSUBOTA** [#], **Yoshihiro TOMITA** [+]

Graduate School of Science and Technology, Kobe University
1-1 Rokkodai, Nada, Kobe 657-8501, Japan

Division of Computer and Information, Advanced Computing Center, RIKEN
2-1 Hirosawa, Wako, Saitama 351-0198, Japan

[*]e-mail: adachi@mech.kobe-u.ac.jp
[†]e-mail: kunimoto@solid.mech.kobe-u.ac.jp
[#]e-mail: tsubota@postman.riken.go.jp
[+]e-mail: tomita@mech.kobe-u.ac.jp

Abstract. *From mechanical viewpoints, stress concentration and stress shielding at the bone and femoral stem interface are believed to cause bone adaptation and loosening in an artificial hip joint system. To attain proper long-term fixation and mechanical compatibility, it is important to take into account the mechanical conditions at this interface in designing the stem shape. In this study, a new method for designing the femoral stem shape using voxel finite element model is proposed based on stress uniformity at the bone-stem interface. The basic features of the proposed design method were first investigated by using an idealized simple femur-stem model in terms of the initial stem shape and fixed conditions at the interface. The method was then applied to the design of the stem shape by using a realistic femur-stem model based on digital images such as those obtained from X-ray CT slice data for a femur and CAD data for a stem. These design simulation studies revealed that the initial stem shape and the fixed conditions at the bone-stem interface were key factors in designing the stem shape based on stress uniformity at the interface. It was also found that actual bone/stem shapes and loading conditions should be considered in the stem shape design that could be realized by using digital image-based modeling and simulation techniques.*

1. INTRODUCTION

Bone adaptation and loosening around an artificial hip joint stem are closely related to local mechanical conditions at the bone-stem interface such as the stress concentration and stress shielding [1]. To attain proper long-term fixation and mechanical compatibility, it is important to take into account the mechanical conditions at the interface in shape design of a femoral stem. The shape of the stem has so far been mainly designed based on an experimental evaluation and clinical results [2]. However, due to the difficulty in experimentally assessing the stress concentration and stress shielding, it is desired to establish a novel method for designing the stem shape that is assisted by computational modeling and simulation techniques.

We report in this study the application of trabecular surface remodeling simulation method [3] for shape modification as a new method to design the stem shape that is based on stress uniformity at the bone-stem interface. This method involves modeling the stem and femur as an assemblage of voxel finite elements, and then modifying the stem shape by adding/removing the voxel elements to/from the stem surface based on stress nonuniformity at the interface. The basic features of the proposed design method are first investigated by using an idealized simple bone-stem model. The method is then applied to a realistic femur-stem model based on digital images obtained from CT slice data for a femur and CAD data for a stem.

2. SIMPLE BONE-STEM MODEL STUDY

The basic features of the proposed method were investigated in terms of the initial stem shape and the fixed conditions at bone-stem interface by using two simple bone-stem models with same stem volume, cases S and L, as shown in Figs.1 and 2. The case S stem, a column of 16 mm in diameter and in 32 mm length, was inserted to 3/4 of its length, as shown in Fig.1. The case L stem, a rod of 15 mm in diameter with the mid-section reduced to 5 mm in diameter, and 90 mm length, was inserted into a bone cavity as shown in Fig. 2. To investigate the effect of the fixed region on the stem shape design, fixed and non-fixed interfaces were specified by a given conditions. The non-fixed conditions were applied from the base of the stem to a height of 68 mm (case S) and 70 mm (case L) by inserting dummy elements with low stiffness at the interface.

Assuming linear elasticity, Young's modulus E and Poisson's ratio ν for the materials were set as E_b = 20 GPa and ν_b = 0.30 for the bone, E_s = 200 GPa and ν_s = 0.29 for the stem, and E_d = 0.01 GPa and ν_d = 0.30 for the dummy elements. As boundary conditions, the lower end of the bone was fixed and shearing force F_Y = 300 N was applied in the X direction to the top of the stem. The equivalent stress was calculated under these conditions, and is plotted as a gray-scale image in Figs.1(b) and 2(b). Two model parameters in the rate equation to modify the stem shape [3], threshold values $|\Gamma|$ and sensing distance l_L, were set as $|\Gamma_u - \Gamma_l|$ = 0.8 and l_L = 10.0 mm. The design region was set within the bone cavity and a stem except for the hatched regions shown in Figs.1(b) and 2(b). The total volume of the stem was kept constant during shape modification.

The equivalent stress distribution in the X-Z section and the stem shape obtained by the proposed method for cases S and L are shown in Figs.3(a) and (b). In case S, the stem shape did not grow downward and stayed in the fixed region as shown in Fig.3(a). This was due to the fact that the newly grown stem part was cut off, because the force transmitted through the non-fixed interface was small even if the stem grew downward into the non-fixed region. In case L, the stem shape changed and stayed within the non-fixed region with almost constant length held. This is attributed to the stress distribution at the lower interface caused by the moment due to the long stem shape.

It was found from these studies that the stem shape determined by the proposed design method based on stress uniformity at the bone-stem interface greatly depended on the fixed region that was set as a design condition, as well as on the initial stem shape. This suggests that, for example, arrangement of a porous-coated surface in region where the strong bonding is required is very important in the design of a cement-type stem.

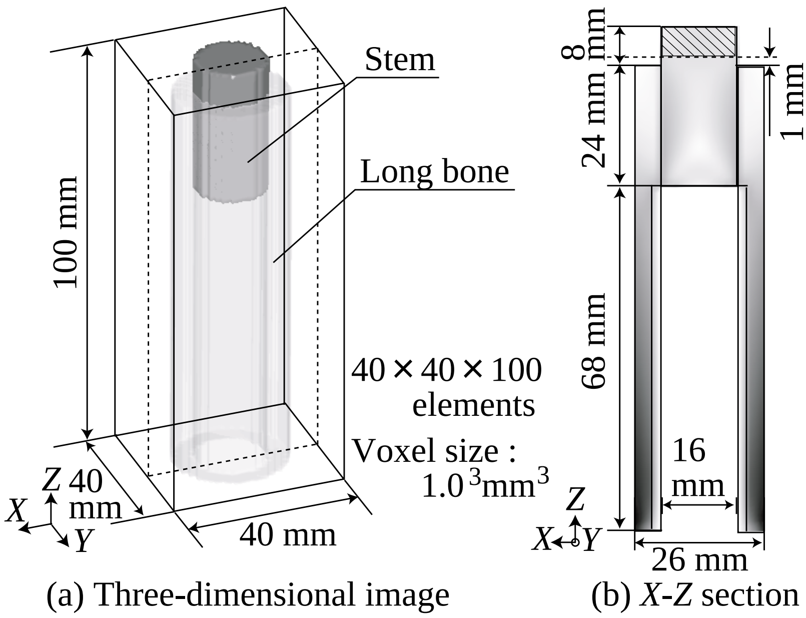


Figure1: Simple model of the bone and short stem (case S).

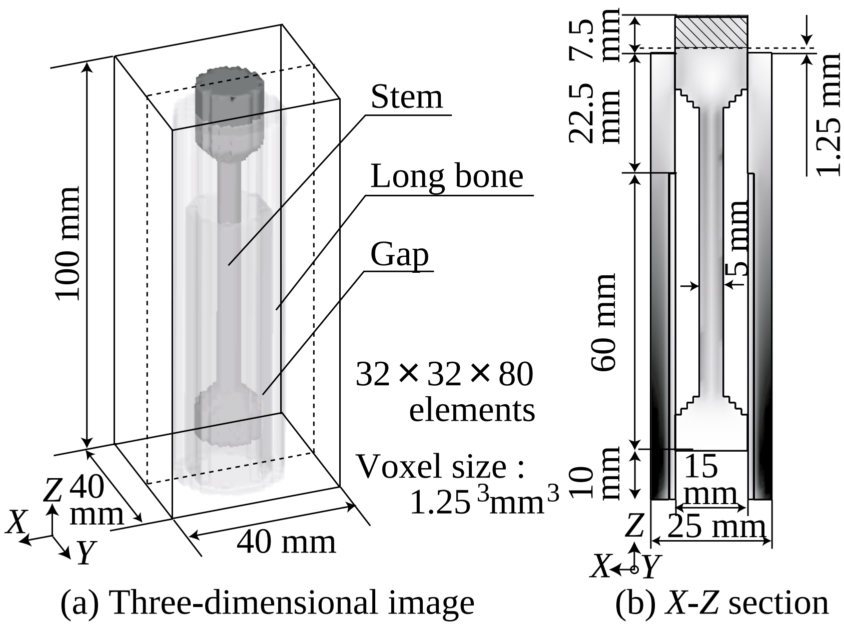
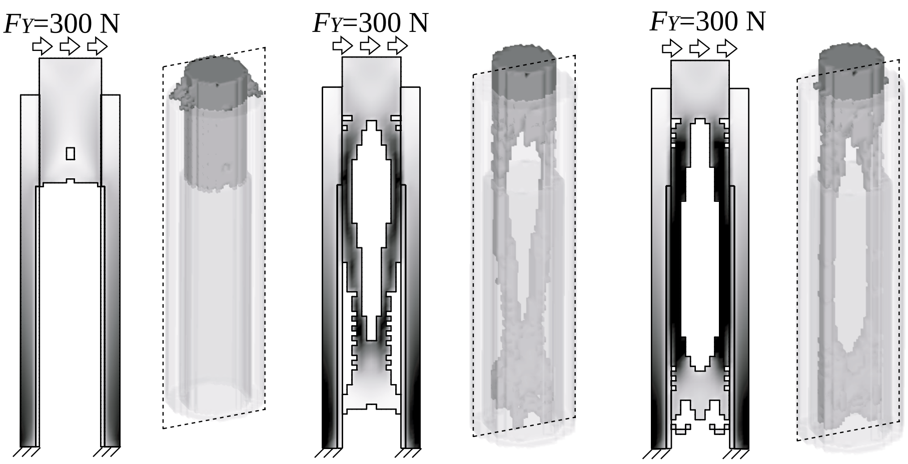


Figure 2: Simple model of the bone and long stem (case L).



(a) Case S (b) Case L (Case L_L) (c) Case L_H

Figure 3: Stem shape and equivalent stress distribution determined by the proposed method.

To examine the effect on stem shape design of the fixed condition at the bone-stem interface, stiffness E_d of dummy elements in the model for case L was changed as a parameter, giving E_d = 0.01 GPa for case L_L and E_d = 0.10 GPa for case L_H. The same design region, material parameters, and boundary conditions, as those for case L in the previous section, were set.

The equivalent stress distribution in the *X-Z* section and stem shapes for cases L_L and L_H obtained by the proposed method are shown in Figs.3(b) and (c). In case L_L, a higher stress was distributed in the upper region of the stem because of the low stiffness of the dummy elements; on the contrary, a higher stress was distributed in the lower region in case L_H because of the moment due to the high stiffness of the dummy elements. This difference in dummy element stiffness gave rise to different stem shapes determined by the proposed method: that is, the stem shape was divided into two parts from the upper region in case L_L, and from the upper and lower regions in case L_H.

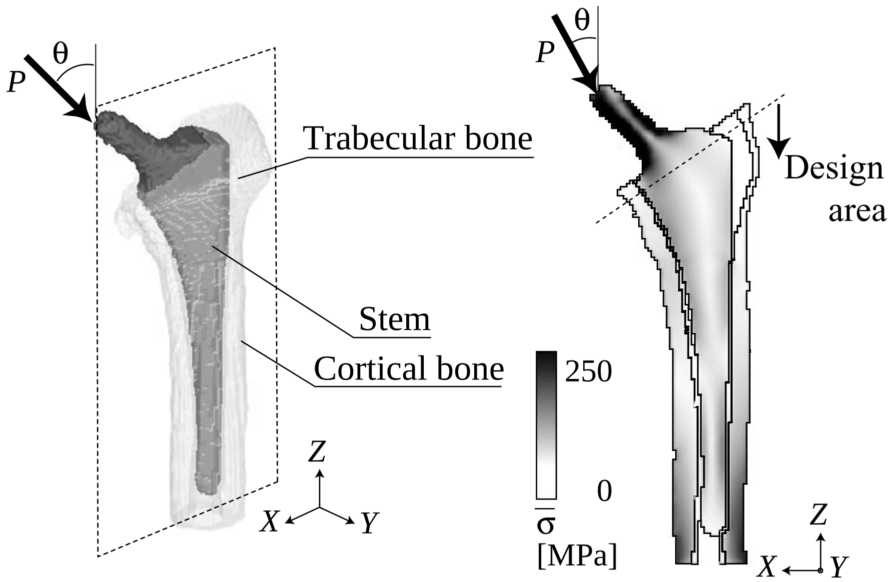
The results of these studies revealed that the pattern of force distribution due to the moment changed depending on the stiffness of the dummy elements, yielding different stem shapes. This suggests that the fixed conditions at the bone-stem interface have to be considered in the stem shape design. For example, the thickness and mechanical properties of the cement and cancellous bone are very important in the shape designs of the cement-type stem and cement-less-type stem, respectively.

3. DIGITAL IMAGE-BASED MODEL STUDY

Based on the results of the previous study with a simple model, the importance from a mechanical viewpoint of the initial stem shape and the fixed condition at the bone-stem interface was revealed in the shape design of a stem. The proposed method is applied in this section to a realistic femur-stem model.

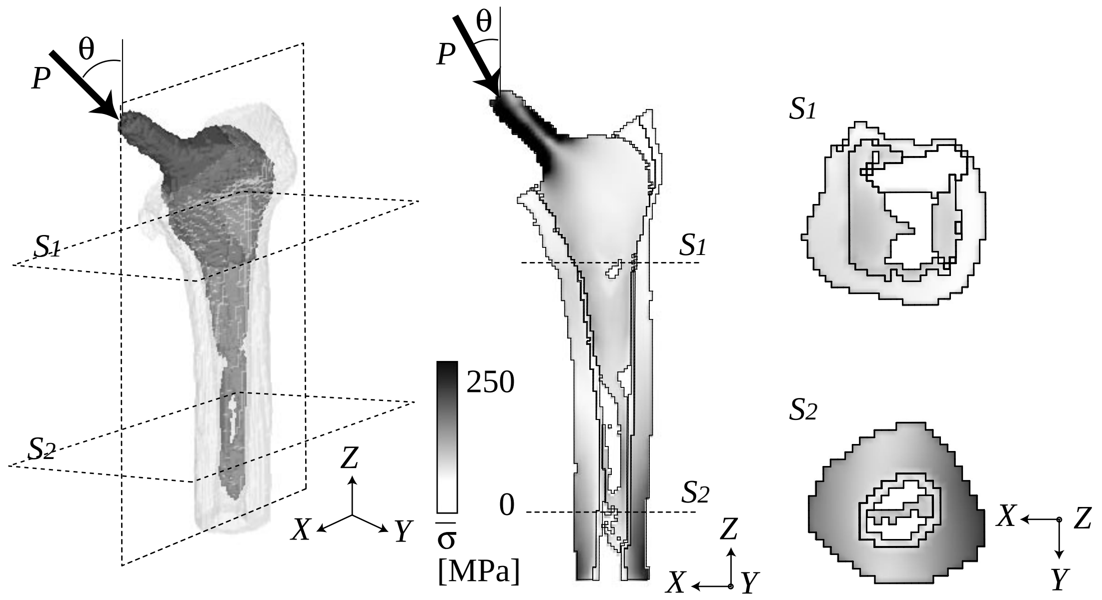
The femur-stem model was constructed based on digital images obtained from CT slice data (femur) and CAD data (stem), as shown in Fig.4(a). The size of each finite element was 527 μm, and the total number of elements was 84,075. Fixed and non-fixed conditions were respectively applied to the stem at the cancellous and cortical bone interfaces. Assuming linear elasticity, the material properties were set as E_c = 20 GPa and ν_c = 0.30 for cortical bone, E_t = 0.2 GPa and ν_t = 0.30 for cancellous bone, E_s = 200 GPa and ν_s = 0.29 for the stem, and E_d = 0.01 GPa and ν_d = 0.30 for the dummy elements. As boundary conditions, the lower end of the femur was fixed and a single loading corresponding to that in the one-legged stance was applied, *i.e.*, P = 2317 N at angle θ = 24 deg in *X-Z* plane [4], as shown in Fig.4(a). Under these boundary conditions, the equivalent stress was calculated as shown in Fig.4(b). Two model parameters were set as $|\Gamma_u - \Gamma_l|$ = 0.6 and l_L = 6.0 mm, respectively. Assuming insertion of the stem into the marrow cavity, the design region was set in the cavity below the dotted line shown in Fig.4(b), the total volume of the stem being kept constant during shape modification.

The stem shape and equivalent stress distribution in the *X-Z* and *X-Y* sections for the femur-stem model obtained by the proposed method are shown in Fig.5. In the proximal region, compressive loading from the top of the stem to the medial cortical bone yielded a bending moment in the femoral neck region, compressive stress in the medial region and tensile stress in the lateral region. This stress distribution due to the bending moment resulted in and increased stem thickness in the lateral and medial regions, as shown in section S_1 in Fig.5(b). In the distal region, a cavity was generated at the base of the stem, as shown in section S_2 in Fig.5(b). As a result of the stem shape modification, stress nonuniformity on the stem surface, measured by the standard deviation of the element equivalent stress, was decreased toward uniform distribution as shown in Fig.6(a), indicating that the stress distribution had been regulated to become uniform. In addition, the total strain energy in the stem was decreased, indicating that the stiffness of the stem had been increased by changing its shape. Although the obtained stem shape is complicated, it reflects realistic mechanical conditions.



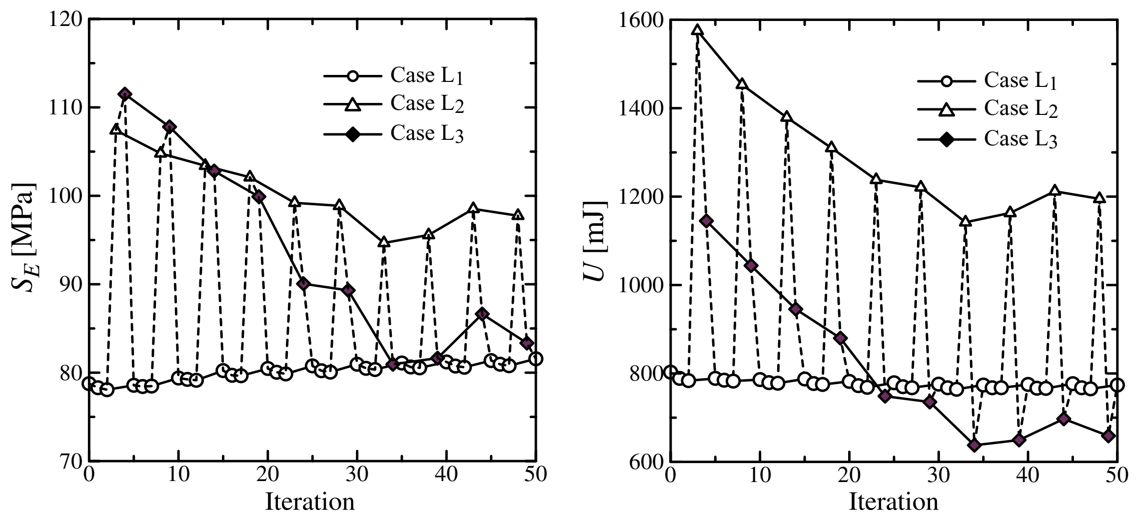
(a) Three-dimensional image (b) *X-Z* section

Figure 4: Initial stem shape and equivalent stress distribution in the image-based femur-stem model



(a) Three-dimensional image (b) *X*-*Z* and *X*-*Y* section

Figure 5: Stem shape and equivalent stress distribution in the image-based femur-stem model determined by the proposed method



(a) Standard deviation of equivalent stress in stem (b) Total strain energy in stem

Figure 6: Progressive changes in the equivalent stress and strain energy in the stem during the iterative design process

4. CONCLUSION

The results of these design simulation studies reveal that the initial stem shape and the fixed conditions at the bone-stem interface are key factors in the design of a stem shape based on stress uniformity at the interface. In addition, it was found that the actual femur/stem shapes and the loading conditions should be considered in the design of the stem shape that could benefit from the use of digital image-based modeling and simulation techniques.

5. REFERENCES

[1] Weinans, H., Sumner, D. R., Igloria, R., and Natarajan, R. N., Sensitivity of periprosthetic stress-shielding to load and the bone density-modulus relationship in subject-specific finite element models, Journal of Biomechanics, Vol. 33, No. 7, pp. 809-817 (2000).

[2] Murray, D. W., Carr, A. J., and Bulstrode, C. J., Which primary total hip replacement?, Journal of Bone and Joint Surgery, Vol. 77B, No. 4, pp. 520-527 (1995).

[3] Adachi, T., Tsubota, K., Tomita, Y., and Hollister, S. J., Trabecular surface remodeling simulation for cancellous bone using microstructural voxel finite element models, The Transaction of ASME, Journalof Biomechanical Engineering, Vol. 123, No. 5, pp. 403-409 (2001).

[4] Beaupre, G. S., Orr, T. E., and Carter, D. R., An approach for time-dependent bone modeling and remodeling-application: a preliminary remodeling simulation, Journal of Orthopaedic Research, Vol. 8, No. 5, pp. 662-670 (1990).