Multiscale stress analysis of trabecular bone around acetabular cup implant by finite element mesh superposition method

Makoto TSUKINO*, Naoki TAKANO**, Adrien MICHEL*** and Guillaume HAÏAT***

Quint Corporation
1-14-1 Fuchu-cho, Fuchu, Tokyo 183-0055, Japan
E-mail: tsukino@quint.co.jp

Department of Mechanical Engineering, Keio University
3-14-1 Hiyoshi, Kohoku-ku, Yokohama 223-8522, Japan

Université Paris-Est Créteil Val-de-Marne
61 avenue du général de Gaulle 91010 Créteil Codex, France

Received 1 July 2015

Abstract
Microscopic stress was calculated with 0.017 mm resolution in a macroscopic model with approximately 100 mm size using the finite element mesh superposition method. To bridge the large gap in resolution, an intermediate finite element model was newly used. This multiscale computational procedure was applied to the biomechanical problem to analyze the microscopic stress in the trabecular bone around acetabular cup implant in total hip arthroplasty, which occurs by direct contact of the implant with trabecular bone. In the microstructural modeling of highly porous media such as the trabecular bone, special attention was paid to the boundary of microstructure model for both homogenization procedure and mesh superposition procedure. Three demonstrative numerical results revealed that higher stress occurred at microscale due to macroscopic stress concentration, which is hardly estimated by only bone volume fraction.

Key words: Finite element mesh superposition method, Multiscale analysis, Biomechanics, Porous trabecular bone, Homogenization, Total hip arthroplasty

1. Introduction

The use of endosseous implant such as total hip arthroplasty (THA), total knee arthroplasty (TKA) or oral implant is now increasing, with the help of development of new materials and the advancement of implant design to achieve long term stability based on various biomechanical studies (Zivkovic, et al., 2010, Haïat, et al., 2014). Finite element analyses (FEA) have contributed to understand biomechanical phenomena occurring in-vivo as well as to improve implant design (Witzel, et al., 2008, Kluess, et al., 2009). The load transfer from metal implant to bone is an important issue in biomechanics (Mueller, et al., 2009). Since bone consists of dense cortical bone and inner porous trabecular bone (or cancellous bone) with very complex 3D microarchitecture, it is important to consider the peri-implant trabecular bone (Basler, et al., 2013).

In order to take the microstructure of trabecular bone into account in FEA, micro-CT image-based modeling for cadaver is an efficient technique (Hollister and Kikuchi, 1994). One of the authors have so far analyzed the peri-implant microscopic stress for jaw bone in the dental biomechanics field (Ohashi, et al., 2010). However, for artificial joint such as THA and TKA, no numerical study has been reported on the microscopic stress analysis in trabecular bone including metal endosseous implants. So far, macroscopic homogenized models have been used in FEA using empirical prediction rule with respect to bone density. One of the problems of using the empirical rule is that the anisotropic/orthotropic nature of the trabecular bone is not considered. The difficulty lies in the large gap in resolution...
to model both trabecular microstructure and metal joint, which results in a huge number of finite elements.

To solve this problem, this paper utilized the multiscale computational methods, i.e., asymptotic homogenization method (Lions, 1981, Guedes and Kikuchi, 1990) and finite element mesh superposition method (or s-version FEM) (Fish, 1982, Takano, et al., 1999, Takano and Okuno, 2004) to analyze microscopic stress around acetabular cup implant in THA. The microstructure model of trabecular bone was obtained by micro-CT with 0.017 mm resolution. The orthotropic homogenized model was obtained using a homogenization method and used in the macroscopic model with a representative dimension of 100 mm. Under the load transferred from the implant, the microscopic stress under non-uniform strain condition was calculated by finite element mesh superposition method. The large gap in resolution was found to be a problem, and therefore an intermediate model was used to bridge macro- and micro-scales.

2. Object and purpose of analysis

This study focuses on an acetabular cup (AC) implant used in THA as illustrated in Fig. 1(a). It is made of CoCr and its diameter is approximately 30 mm. In the surgical protocol, after eliminating the skin layer of the bone, i.e., cortical bone and realization of a cavity, the AC implant is impacted into bone tissue (Mathieu, et al., 2013, Michel, et al., 2015). Then, it is directly in contact with trabecular bone. The typical microstructure of trabecular bone can be seen in the micro-CT image shown in Fig. 1(b). The representative dimension of the trabecular strut is approximately equal to 0.1 mm. The resolution of micro-CT image is 0.017 mm to obtain the clear image of microstructure.

![Acetabular cup made of CoCr with approximately 30 mm diameter](image1)

![Micro-CT image of trabecular architecture](image2)

Fig. 1 Acetabular cup implant in total hip joint arthroplasty

The purpose of this study is to build a computational method to analyze the microscopic stress in the trabecular strut. Figure 2 shows a conceptual model of cortical bone (red) and trabecular bone (yellow) around AC implant (blue). The interface of dissimilar material is assumed to be perfectly bonded. The metal cup, trabecular bone region and cortical bone have the thickness of 4 mm, 21 mm and 3 mm, respectively. Hence, the gap in length between the thickness of trabecular bone region and the resolution to express the microstructure, 0.017 mm, is over 1,200.

The edge of the cortical bone on the top surface in Fig. 2 was assumed to be fully constrained. Variety of loading conditions must be analyzed to simulate the in-vivo situation, but as shown in Fig. 2(b), only vertical load was applied in this paper. To set the orthotropic characteristics of the trabecular architecture, maximum macroscopic Young’s modulus was assumed to be in the vertical direction as shown by arrows in yellow region in Fig. 2(b). The macroscopic material model was calculated by homogenization method, whose reliability has been validated for engineering materials (Takano, et al., 2003) as well as human vertebral trabecular bone (Yoshiwara, et al., 2011).

The macroscopic strain is distributed in the trabecular bone region. To calculate the microscopic stress under this condition, the homogenization method is useless because of the non-uniformity of macroscopic strain. Instead, the finite element mesh superposition method is used together with the homogenized model. It was firstly proposed by Fish (Fish, 1992) for mesh refinement (Takano, et al., 1999, Park, et al., 2003), and applied to the multiscale problems for heterogeneous materials, local crack, inclusion and interface problems (Takano, et al., 2000, Okada, et al., 2005).
Moreover, one of the authors have proposed a new technique to use this method in combination with a homogenization model (Takano and Okuno, 2004, Kawagai, et al, 2006). This technique seemed to be applicable also to the current problem. However, the very large gap of mesh size between macro- and microscopic models in this study led to the idea of using intermediate resolution model.

In the following chapters, the formulation of finite element mesh superposition method and the key point in the modeling of porous media are described concisely. In the current analysis, emphasis is put on the use of intermediate resolution model.

### 3. Computational method

In the global model using homogenized material model for trabecular bone region (Fig. 2(a)), 1 mm sized finite elements are used. On the other hand, to calculate the microscopic stress in the trabecular bone, a local model whose element size is 0.017 mm is superimposed onto the global model.

Let the whole domain be \( \Omega = \Omega^G \cup \Omega^L \) as shown in Fig. 3(a), where the local model occupies \( \Omega^L \). Denote the nodal displacement vectors calculated by global mesh as \( \{u^G\} \) and that by local mesh as \( \{u^L\} \). In \( \Omega^L \), the displacement \( \{u^L\} \) is defined by
\[
\{u\} = \{u^G\} + \{u^L\}
\]
To assure the continuity of the displacement at the boundary \( \Gamma^{GL} \), the following condition must hold.
\[
\{u^L\} = \{0\}
\]

When the voxel mesh is used for both global and local models, the advantage of mesh superposition is outstanding if a different parallelepiped mesh is used for the global and local models as shown in Fig. 3(b). In Chapter 4, the local model is rotated because the micro-CT imaging direction is hardly controlled to agree with the macroscopic coordinate system. From Eq. (1), the strain is defined as follows using strain-displacement matrix \( [B] \).

---

Fig. 2 Multiscale finite element analysis model

(a) Top view  (b) Side view

Fig. 3 Superposition of local model onto global model

(a) Definition of global domain and local domain  (b) Mesh superposition in case of voxel mesh

\[ \{ \varepsilon \} = [B][\mu]\{ u \} = \left( [B^G][\mu^G] + [B^L][\mu^L] \right) \text{ in } \Omega^L \]
\[ \text{in } \Omega^G \]  
(3)

Note that the strain is discontinuous at boundary \( \Gamma^{GL} \).

In the asymptotic homogenization method, the displacement is defined by the macroscopic term \( \{ \mu^0 \} \) and the perturbed term \( \{ \mu^1 \} \) with a scale ratio \( \lambda \) between macro- and microscales as follows.

\[ \{ u \} = \{ \mu^0 \} + \lambda \{ \mu^1 \} \]  
(4)

The finite element mesh superposition method does not use the scale ratio shown in Eq. (4), but instead the dimension is explicitly expressed by the mesh size. The drawbacks of the asymptotic homogenization method are that it is impossible to consider the real dimensions of microstructure and that the microscopic stress can be calculated only under uniform macroscopic strain field. These problems were resolved by the finite element mesh superposition method. This is a sort of strongly coupled method to solve global and local problems as shown in the following stiffness equation.

\[
\begin{bmatrix}
K^G \\
K^{GL} \\
K^{L}
\end{bmatrix} 
\begin{bmatrix}
\{ \mu^G \} \\
\{ \mu^L \} \\
\{ \mu^G \}
\end{bmatrix} = 
\begin{bmatrix}
\{ f \} \\
\{ 0 \}
\end{bmatrix}
\]  
(5)

In the problem shown in Fig. 2, stress-strain matrices for the acetabular cup implant, cortical bone, trabecular bone and homogenized model of trabecular bone region are denoted by \( [D_{AC}] \), \( [D_{co}] = [D_{mc}] \) and \( [D_{tr}], \) respectively. The submatrix \( [K^G] \) is derived as

\[
[K^G] = \int_{\Omega^G} [B^G]^T [D^G] [B^G] d\Omega^G + \int_{\Omega^L} [B^G]^T [D^L] [B^G] d\Omega^L
\]  
(6)

where \( [D^G] = [D_{AC}] \) or \( [D_{co}] \) or \( [D_{mc}] \) and \( [D^L] = [D_{tr}] \) or \( [0] \). Since the resolution of global model and that of local model is much different, the second term of right hand side of Eq. (6) can be approximated by the following equation.

\[
[K^G] \approx \int_{\Omega^G} [B^G]^T [D^G] [B^G] d\Omega^G + \int_{\Omega^L} [B^G]^T [D^L] [B^G] d\Omega^L
\]  
(7)

In the microstructure modeling of highly porous media such as the trabecular bone, special attention must be paid to the boundary conditions of the model. In the homogenization method, to assure the setting of periodic boundary condition to the unit cell model for homogenization, the microstructure model must be wrapped by a hypothetical very soft material as shown in Fig. 4(b). Whilst in the finite element mesh superposition method, the local model to be superimposed on the global model must be wrapped by the homogenized material model that is used in the global model as shown in Fig. 4(c) (Kawagai, et al., 2006). Note that, in Fig. 4, the gray colored part shows the trabecular microstructure in this analysis.

Let the stress-strain matrix of this wrapping layer for mesh superposition method be \( [D_{wrap}] = [D_{tr}] \). Then, Eq. (7) can be written simply as following (Takano and Okuno, 2004):

![Fig. 4 Finite element models of highly porous microstructure](image-url)

(a) Microstructure region of interest

(b) Unit cell model for homogenization analysis

(c) Local model for mesh superposition analysis
Please see ref. (Takano, et al., 2000, 2001) for detailed numerical procedure.

One problem lies in the very large gap in mesh size between global and local models as mentioned before. A preliminary test with very large resolution gap revealed that no accurate results were obtained. Therefore, in this paper, an intermediate model is used in the finite element mesh superposition method. By using an intermediate model with 1/3 mm element size, the convergence of the accuracy has been confirmed. The physical quantities in the intermediate model were obtained by solving a Dirichlet problem where the nodal displacements on boundary surface are provided from global analysis. Finally, the mesh superposition method was applied to the intermediate and local models.

4. Numerical result and discussion

The homogenization analysis was carried out to obtain the macroscopic homogenized properties of trabecular bone based on the micro-CT images. Figure 5 shows the 3D unit cell model. Voxel elements were used, and the number of elements is equal to 11,202,142 in Fig. 5(a) and that of Fig. 5(b) with wrapping elements is equal to 12,387,577. The coordinate system $x' - y' - z'$ was defined temporarily based on the micro-CT imaging direction. The bone volume fraction in the unit cell model is 10.6 %. Isotropic material model is used for the bone tissue, whose Young’s modulus and poisson’s ratio are 15 GPa and 0.3 (Yoshiwara, et al., 2011). Those of wrapping elements are 0.015 MPa and 0.3, respectively.

The homogenized elastic tensor calculated in $x' - y' - z'$ coordinate system in Fig. 5(a) showed strongly orthotropic condition. The orthotropy of the trabecular bone is natural and can be seen also in vertebra and so on, to support the primary external load. In vertebral trabecular bone, the maximum Young’s modulus is in the vertical direction to support the self-weight [Yoshiwara, et al., 2011]. Also in this analysis, the primary orthogonal direction of the trabecular bone which provides the maximum Young’s modulus, $z$ direction in Fig. 5(a), was firstly calculated. The Young’s moduli in $x$ and $y$ were much closer and approximately 1/3 of that in $z$ direction. The correlation between $x - y - z$ and $x' - y' - z'$ coordinate systems was obtained as follows.

\[
\begin{pmatrix}
0.227 & -0.839 & 0.495 \\
0.726 & -0.193 & -0.660 \\
0.649 & 0.509 & 0.565
\end{pmatrix}
\begin{pmatrix}
x \\
y \\
z
\end{pmatrix}
= 
\begin{pmatrix}
x' \\
y' \\
z'
\end{pmatrix}
\]  

In the same way with the vertebra, it is natural that the vertical loading given from the acetabular cup implant is assumed to be the primary load as shown in Fig. 2. For this reason, the maximum Young’s modulus was assumed to be in $z$ axis in Fig. 5(a) and Fig. 2(b). In the later analysis by mesh superposition method, the macroscopic stress-strain matrix was used as rotated instead of the engineering constants such as Young’s modulus.
Then, the macroscopic analysis was carried out under the loading and constraint conditions as described in Chapter 2. Figure 6 shows the cross-sectional view of Mises stress distribution. Due to the orthotropic nature of the trabecular bone, the distribution is not symmetric.

Fig. 6 Macroscopic Mises stress distribution and location to analyze microscopic stress, A, B and C, by finite element mesh superposition method because the macroscopic stress and strain are not uniform in the local position.

The microscopic stress was analyzed at three locations as shown in Fig. 6. Maximum macroscopic Mises stress was seen at the interface edge near location A. It is shown that the macroscopic stress/strain field is non-uniform in the microstructure model. Hence, finite element mesh superposition is applied to calculate the microscopic stress.

Figure 7 illustrates three-scale models, i.e., macroscopic, intermediate and local models to calculate the microscopic stress at location B in Fig. 6. The voxel mesh sizes for three models are shown in Fig. 7. The intermediate model has the dimension of approximately 10 mm x 17 mm x 14 mm and consists of 64,260 voxel elements.

The microscopic Mises stress is more than 10 times larger than macroscopic Mises stress. Higher microscopic...
stress was seen in the left hand side of the local model, which is closer to the interface edge in the macroscopic model. The stress distribution in the local model is not periodic. The average microscopic stress in the local model was 1.9 MPa. Considering that the bone volume fraction is 10.6 %, much higher stress was predicted due to the macroscopic stress concentration by means of finite element mesh superposition method.

Calculated microscopic Mises stresses at three locations were compared as the cumulative histogram as shown in Fig. 8. The stress at location A is much higher because it is close to the interface edge. Such comparison by histogram of stress distribution is more reasonable and confident than comparison by maximum point-wise stress values. To find possible higher microscopic stress in the trabecular bone region, more number of mesh superposition analyses must be performed. However, through above three analyses, the effectiveness of the computational procedure using finite element mesh superposition method with homogenized material model and intermediate finite element model has been demonstrated.

![Microscopic Mises stress histogram](image)

Fig. 8 Comparison of microscopic Mises stress at 3 locations, A, B and C indicated in Fig. 6 by means of stress histogram, which shows the microscopic stress at location A shows much higher values compared to other 2 locations

5. Conclusion

Aiming at the numerical analysis of microscopic stress in the trabecular bone around acetabular cup (AC) implant in total hip arthroplasty, which occurs by direct contact of metal implant with trabecular bone, multiscale modeling and computational procedures have been presented. In the framework of published methodology using asymptotic homogenization method and finite element mesh superposition method, newly used intermediate finite element model could resolve the problem of large gap in resolution between macro- (global) and micro- (local) models.

Three analyses were presented in this paper under one loading condition. The cumulative histogram was found to be effective to compare the microscopic stresses. Additional microscopic stress analyses will provide higher stress. Also, the effects of the loading conditions will be analyzed, which may give new insight into the implant stability problem. Consideration of inter-individual differences is another interesting issue such as the trabecular network architecture and material properties of bone tissue.

Acknowledgment

The authors are grateful for the help of students of Keio University, Mr. Kyohei Hatano to perform the analyses as well as Mr. Daichi Kurita to write this manuscript. This study has been supported by the Japan Society for the Promotion of Science (JSPS) KAKENHI (Grant-in-Aid for Scientific Research (B), 24360047) and the French National Research Agency (ANR) through the PRTS program (project OsseoWave n°ANR-13-PRTS-0015-02).

References

Guedes, J. M. and Kikuchi, N., Preprocessing and postprocessing for materials based on the homogenization method


