J. KSIAM Vol.15, No.1, 57–65, 2011

THE EFFECTS OF MESH STYLE ON THE FINITE ELEMENT ANALYSIS FOR ARTIFICIAL HIP JOINTS

JAEMIN SHIN[1], DONGSUN LEE[1], SUNGKI KIM[1], DARAE JEONG[1], HYUN GEUN LEE[1], AND JUNSEOK KIM[1†]

[1]DEPARTMENT OF MATHEMATICS, KOREA UNIVERSITY, SEOUL 136-701, REPUBLIC OF KOREA
E-mail address: cfdkim@korea.ac.kr

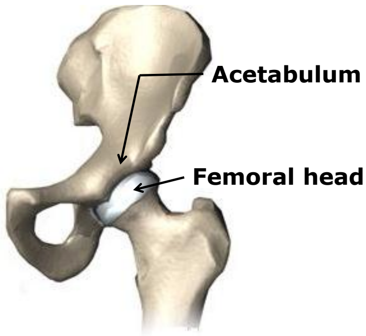
ABSTRACT. In this paper, a good quality mesh generation for the finite element method is investigated for artificial hip joint simulations. In general, bad meshes with a large aspect ratio or mixed elements can give rise to excessively long computational running times and extremely high errors. Typically, hexahedral elements outperform tetrahedral elements during three-dimensional contact analysis using the finite element method. Therefore, it is essential to mesh biologic structures with hexahedral elements. Four meshing schemes for the finite element analysis of an artificial hip joint are presented and compared: (1) tetrahedral elements, (2) wedge and hexahedral elements, (3) open cubic box hexahedral elements, and (4) proposed hexahedral elements. The proposed meshing scheme is to partition a part before seeding so that we have a high quality three-dimensional mesh which consists of only hexahedral elements. The von Mises stress distributions were obtained and analyzed. We also performed mesh refinement convergence tests for all four cases.

1. INTRODUCTION

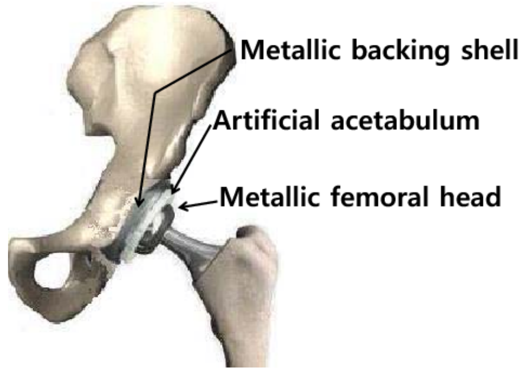
The hip joint is a multiaxial spheroidal joint where the articulating bone surfaces are covered with articular cartilage [7] (see Fig. 1(a)). Hip joint replacement is a surgery to replace all or part of the hip joint with an artificial hip joint (see Fig. 1(b)). The most common reason to have an artificial hip joint operation is to relieve severe arthritis pain that is limiting daily activities. The artificial hip joint is composed of the metallic femoral head, the ultra-high molecular weight polyethylene (UHMWPE) cup, and the metallic backing shell [1].

Because the performances of an artificial hip joint are dependent on the contact stress distribution, understanding stress distribution is very important for designing an optimal artificial hip joint [8]. To calculate accurately the stress distribution on the artificial hip joint, we need a

Received by the editors March 20, 2011; Accepted March 21, 2011.
2000 *Mathematics Subject Classification.* 65M60, 74S05, 74A10.
Key words and phrases. Finite element analysis, von Mises Stress, Artificial hip joint, ABAQUS.
† Corresponding author.



(a)



(b)

FIGURE 1. Schematic diagram of (a) a human hip joint and (b) an artificial hip joint.

good quality mesh. The main aim of the present work is to study the effect of the meshing on the finite element analysis for the artificial hip joint system.

This paper is organized as follows. Section 2 introduces the finite element model including mesh generation. Computational results showing the effect of the meshing are presented in Section 3. Finally, our conclusions are drawn in Section 4.

2. FINITE ELEMENT MODEL

The finite element (FE) analysis is a computational technique widely used to calculate stresses and strains within artificial hip joints. In this work, the FE analysis is carried out using the commercial finite element software ABAQUS (version 6.8) [6]. The artificial hip joint consists of three parts: the metallic backing shell, artificial acetabulum, and metallic femoral head. In our FE model, we consider the interaction between artificial acetabulum and metallic femoral head. Consequently, we only simulate stresses on the contact area between the artificial acetabulum and metallic femoral head. Fig. 2 shows the FE model of the artificial hip joint.

The radius of the femoral head is $14mm$ and the thickness of the artificial acetabulum is $8mm$, which are the standard sizes normally used in the clinic. Several different finite elements were used in ABAQUS simulations of the artificial hip joint. Three basic elements are shown in Fig. 3. Our FE model is comprised of linear brick elements (see Fig. 3(a)). The reason for taking linear brick elements is that hexahedral meshes yield more accurate solutions than their tetrahedral counterparts (see Fig. 3(b)) for the same number of edges [2, 4, 5]. The finite element mesh is a key factor for an accurate and efficient analysis. A mix of elements can carry an extremely high error and also give the wrong stress neighboring elements. Moreover it is difficult to achieve the convergence in the neighborhood of the mixed node [11]. Thus, entire analysis could be failed. Therefore, we need to avoid the mixing of meshes to improve convergence rate and get better simulation results.

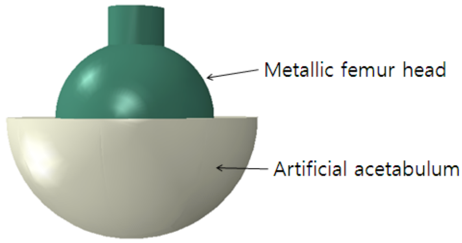
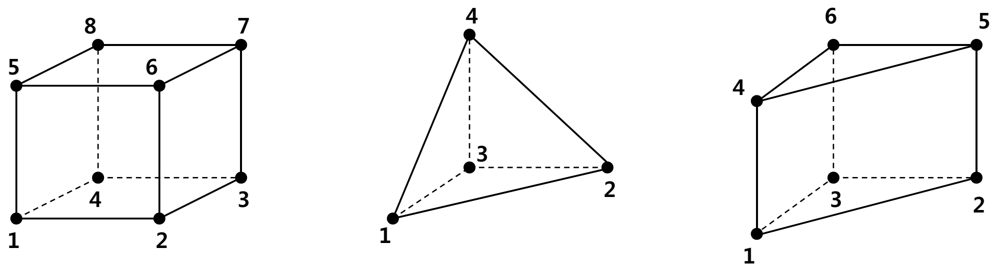


FIGURE 2. The FE model of the artificial hip joint.



(a) Linear brick element (C3D8) (b) Tetrahedral element (C3D4) (c) Wedge element (C3D6)

FIGURE 3. Three different finite elements.

2.1. **Mesh generation procedure.** In generating mesh, the most important factor is to capture the contact surface between the femoral head and the acetabulum cup. The interest region is located at the center of the acetabulum cup core [10]. Figure 4 shows four different meshes on finite element analysis: (a) tetrahedron model, (b) hexahedron with wedge model, (c) hexahedron with open cubic box model, and (d) hexahedron and uniform model. Bottom row shows closeup of each mesh.

The mesh in Fig. 4(c) (we also called the ‘butterfly’ design) is based on an open cube box concept, allows the use of a single element type, and hence avoids a mix of tetrahedral and hexahedral elements and the potential of irregular stress concentrations [3]. However, in mesh generation, the technique generating butterfly model in Fig. 4(c) has the weakness which is more complicated to make mesh than the model in Fig. 4(d).

In this paper, we compare four mesh types (see Fig. 4) of a given acetabulum cup geometry to find a mesh type which has computational accuracy and efficiency for three-dimensional modeling of acetabulum cup. Next, a mesh generating technique of hexahedron model in Fig. 4(d) is described as following.

Step 1: Partitioning into four parts (Fig. 5(b)).

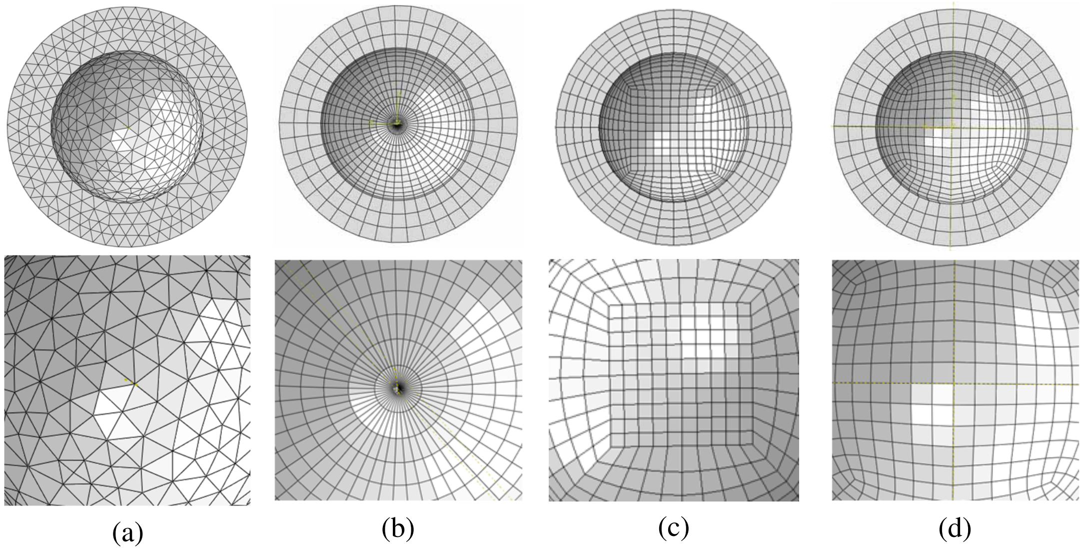


FIGURE 4. Four different meshes on finite element analysis: (a) tetrahedron model, (b) hexahedron with wedge model, (c) hexahedron with open cubic box model, and (d) hexahedron and uniform model. Bottom row shows closeup of each mesh.

Step 2: Seeding onto each part (Fig. 5(c)).
Step 3: Generating mesh (Fig. 5(d)).

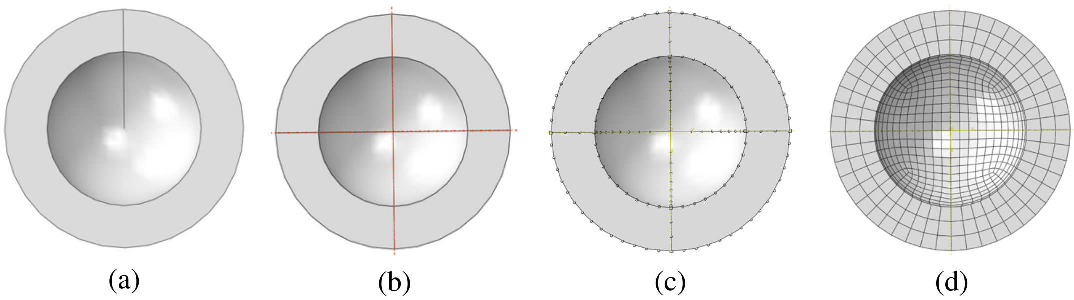


FIGURE 5. (a) Geometry of revolution, (b) partitioning into four part, (c) seeding onto each part, and (d) generating mesh.

2.2. **Penalty method.** The surface-surface contact pair is created between femoral head and acetabulum with contact elements described as contact-penalty method (see Fig.6). Once the contact interface is known we can write the total system energy as

$$
\begin{aligned}
\Pi(u) \quad = \quad & \underbrace{\frac{1}{2}\int_V \epsilon^T D\epsilon dV}_{\text{strain energy}} - \underbrace{\int_v u^T f_v dV}_{\text{body forces}} - \underbrace{\int_S {u_s}^T f_S dS}_{\text{surface forces}} \quad \underbrace{- \sum_C {u_c}^T F_c}_{\text{concentrated forces}} \\
& + \underbrace{\frac{1}{2}\int_S (\epsilon_N (g_{\bar{N}})^2 + \epsilon_T \mathbf{g}_T \cdot \mathbf{g}_T) dS}_{\text{contact forces}},
\end{aligned}
$$

where u is the displacement, ϵ is the strain tensor, D is the stress-strain matrix for the elastic material, f_v is the body forces, f_s is the surface forces, F_c is the concentrated forces, V is the volume, and S is the surface. In the penalty method, the impenetrable surface is considered as a spring and is allowed to penetrate each other. Penalty parameters ϵ_N, ϵ_T and gap functions g_N, $g_{\bar{N}}$, g_T are defined in reference [12].

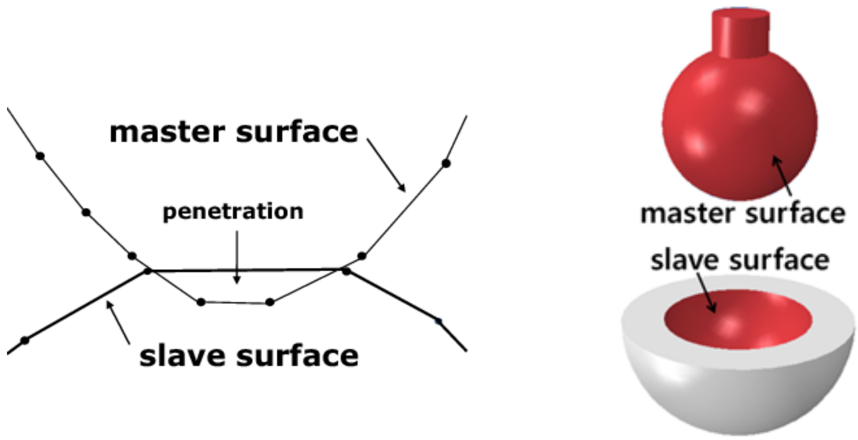


FIGURE 6. (a) Surface-to-surface contact element and (b) finite element model for the contact analysis.

3. COMPUTATIONAL RESULTS

Ti6A14V stainless steel has been used for the metallic femoral head due to their wonderful bio-compatibility behavior in clinical conditions. And the acetabulum cup is made of UHMWPE polymer. Therefore, in this work, the femoral head was assumed to be rigid. A material for acetabulum cup was assumed to be homogeneous, isotropic, and linear elastic solid. The important material parameters required for the FEM analysis are the elastic modulus and the Poisson ratio, which are listed in Table 1 [3]. And the static friction coefficient between two solid surfaces is 0.25.

Materials	Elastic modulus (GPa)	Poisson ratio
UHMWPE	1.4	0.3

TABLE 1. The parameters of artificial hip joint materials.

The quasi-static analysis is used to investigate the meshing effects. We apply a load on the femoral head. The direction of the load is on the rotation axis of the femoral head as shown in Fig. 7(a). In this analysis, the degree of freedom of the femoral head is limited to the direction of the load and backing side of the acetabulum cup is fixed. The loading time is one second and the load is applied gradually over the entire time by using the SMOOTH STEP parameter in ABAQUS (see Fig. 7(b)).

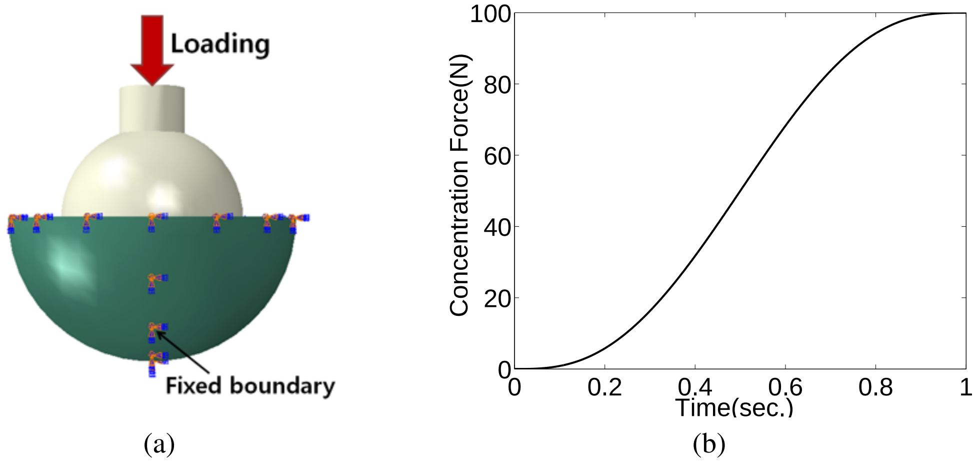


FIGURE 7. (a) The direction of load. (b) Smooth step amplitude definition.

In this paper, we perform numerical tests with different seeding sizes and concentrated forces on four meshes in Fig. 4. The following Tables 2-3 show the von Mises stress results. The von Mises stress is defined as

$$\sigma_\epsilon = \sqrt{\frac{(\sigma_x-\sigma_y)^2+(\sigma_y-\sigma_z)^2+(\sigma_z-\sigma_x)^2+6(\tau_{xy}^2+\tau_{yz}^2+\tau_{zx}^2)}{2}}.$$

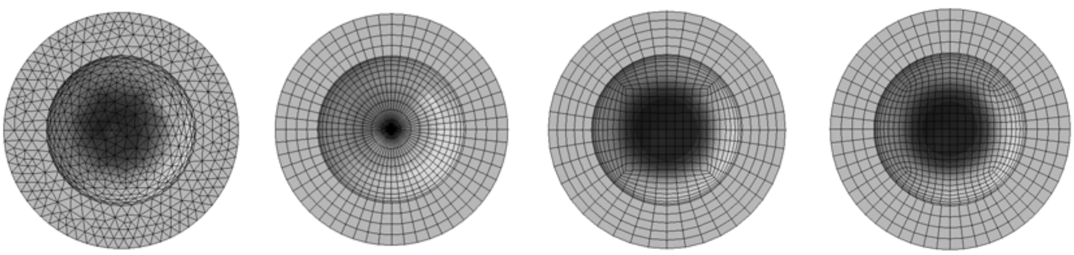
In Table 2, we show four different model results of the maximum von Misses stress (MPa unit) under loading 100N with mesh refinements. First, we can observe that tetrahedron model fails at the seeding size, 1.5. In the case with mixed elements, the results shows an oscillatory behavior with respect to mesh refinements. However, hexahedron with open cubic box and proposed hexahedron models show good convergence results and the values are in good agreement with the values in [9]. Table 3 with 1000N shows similar results. In Fig. 8, we show the von Mises stress contours of the structure with a load 1000N and seeding size 2: (a) tetrahedron mesh, (b) hexahedron mesh with wedge, (c) hexahedron and open cube box mesh, and (d) proposed hexahedron mesh.

Seeding size	Tetrahedron model	Hexahedron with wedge model	Hexahedron with open cubic box model	Proposed hexahedron model
4	0.96513	3.26517	0.87278	0.82816
3	0.84052	0.92625	0.93148	0.89305
2	1.12331	9.29483	0.95506	0.95971
1.5	Deform Error	2.04330	1.03255	1.03339
1.25	1.09540	1.41665	1.07770	1.06093
1	1.20795	1.68110	1.10631	1.09984

TABLE 2. The four model results of the maximum von Misses stress (MPa) under loading with concentration force $100N$.

Seeding size	Tetrahedron model	Hexahedron with wedge model	Hexahedron with open cubic box model	Proposed hexahedron model
4	3.08095	3.48954	3.05187	2.98277
3	3.13320	3.35775	3.18379	3.07369
2	3.53600	12.17590	3.21527	3.23508
1.5	Deform Error	3.92003	3.38127	3.32417
1.25	3.51901	6.48348	3.47091	3.41587
1	3.64780	6.13668	3.49828	3.50807

TABLE 3. The four model results of the maximum von Misses stress (MPa) under loading with concentration force $1000N$.



(a) (b) (c) (d)

FIGURE 8. The von Mises stress contours of the structure with a load 1000N and seeding size 2. (a) Tetrahedron mesh, (b) hexahedron mesh with wedge, (c) hexahedron and open cube box mesh, and (d) proposed hexahedron mesh.

4. Conclusion

In this work, a good quality mesh generation was investigated for artificial hip joint finite element analysis. If it is possible, it is essential to mesh biologic structures with hexahedral elements since hexahedral elements outperform tetrahedral elements during three-dimensional contact analysis. Four meshing schemes for the finite element analysis of an artificial hip joint were presented and compared: (1) tetrahedral elements, (2) wedge and hexahedral elements, (3) open cubic box hexahedral elements, and (4) proposed hexahedral elements. The proposed meshing scheme is to partition a part before seeding so that we have a high quality three-dimensional mesh which consists of only hexahedral elements. We could observe that tetrahedron model failed at the seeding size, 1.5. In the case with mixed elements, the results showed an oscillatory behavior with respect to mesh refinements. However, hexahedron with open cubic box and proposed hexahedron models showed good convergence results. The proposed meshing scheme is simpler than open cubic box model. Therefore, the proposed meshing is a potentially promising technique.

Acknowledgment

This work was supported by the National Research Foundation of Korea(NRF) grant funded by the Korea government(MEST) (No. 2009-0074248).

References

[1] M.R. Abdul-Kadir, U. Hansen, R. Klabunde, D. Lucas, and A. Amis, *Finite element modelling of primary hip stem stability: The effect of interference fit*, Journal of Biomechanics, **41** (3) (2008), 587-594.

[2] S.E. Benzley, E. Perry, K. Merkley, B. Clark, and G. Sjaardama, *A Comparison of all hexagonal and all tetrahedral finite element meshes for elastic and elasto-plastic analysis*, Proceedings of 4th International Meshing Roundtable, Albuquerque, October (1995), 179-191.

[3] S.L. Bevill, G.R. Bevill, J.R. Penmetsa, A.J. Petrella, and P.J. Rullkoetter, *Finite element simulation of early creep and wear in total hip arthroplasty*, Journal of Biomechanics, **38** (12) (2005), 2365-2374.

[4] R. Biswas and R.C. Strawn, *Tetrahedral and hexahedral mesh adaptation for CFD problems*, Applied Numerical Mathematics, **26** (1-2) (1998), 135-151.

[5] A.O. Cifuentes and A. Kalbag, *A performance study of tetrahedral and hexahedral elements in 3-D finite element structural analysis*, Finite Elements in Analysis and Design, **12** (3-4) (1992), 313-318.

[6] K. Hibbitt, *ABAQUS/Standard User's Manual (version 6.8)*, Hibbitt, Karlsson & Sorensen, Inc., USA, 1997.

[7] V. Kralj-Iglic, M. Daniel, and A. Macek-Lebar, *Computer determination of contact stress distribution and size of weight bearing area in the human hip joint*, Computer Methods in Biomechanics and Biomedical Engineering, **5** (2) (2002), 185-192.

[8] F. Liu, I. Lesliea, S. Williams, J. Fisher, and Z. Jin, *Development of computational wear simulation of metal-on-metal hip resurfacing replacements*, Journal of Biomechanics, **41** (3) (2008), 686-694.

[9] K. Rami, K. Arto, K. Yrjo, S. Seppo, and L. Reijo, *The effect of geometry and abduction angle on the stresses in cemented UHMWPE acetabular cups-finite element simulations and experimental tests*, BioMedical Engineering OnLine, **4** (32) (2005) 1-14.

[10] S.H. Teoh, W.H. Chan, and R. Thampuran, *An elasto-plastic finite element model for polyethylene wear in total hip arthroplasty*, Journal of Biomechanics, **35** (3) (2002), 323-330.

[11] M. Tur, J. Fuenmayor, A. Mugadu, and D.A. Hills, *On the analysis of singular stress fields Part 1: finite element formulation and application to notches*, The Journal of Strain Analysis for Engineering Design, **37** (5) (2002), 437-444.

[12] P. Wriggers and T.A. Laursen, *Computational Contact and Impact Mechanics*, Springer-Verlag, Berlin and Heidelberg, 2002.