Research Article

Ikhsan, Joko Triyono*, Aditya Rio Prabowo, and Jung Min Sohn

Investigation of Meshing Strategy on Mechanical Behaviour of Hip Stem Implant Design Using FEA

https://doi.org/10.1515/eng-2020-0087
Received Jan 06, 2020; accepted Jun 05, 2020

Abstract: Hip joint is an important human joints system. The damaged hip joints need to be replaced with artificial hip joints. The Study of the hip joint is very costly therefore another calculation method is demanded to produce good result in acceptable time and cost. Considering this problem, a series of study to assess hip joint performance is conducted using numerical approach. Important parameter for example applied materials are used in the modelling by idealizing Ti-6Al-4V compared to SS 316 L, and stem length was chosen to be 128 mm. ANSYS software was used to analyze models, and designed element size variations were set to be in range 1 to 2.5 mm. The magnitude of force was placed on the femoral head with an angle of 16° from the vertical axis. Results showed that SS 316 L material has smaller deformation than Ti material. Whereas Central Processing (CP) time decreases in increasing element size for both materials. In addition, more variations in mesh size are needed to get more accurate convergent results.

Keywords: Finite Element Analysis, Meshing, Hip Stem, Ti-6Al-4V, SS 316 L

1 Introduction

Hip joint is an important human system which connects the femur and pelvis. Hip joint consists of three main parts, namely: femur, femoral head, and rounded socked. The hip is one of most common joints that are most likely to be replaced during our lifetime due to wear and tear of cartilage, bone degeneration and other factors where weight bearing is a factor [1]. The most common reason to have an artificial hip joint operation is to relieve severe arthritis pain that is limiting daily activities [2]. Hip joint damages require replacement with an artificial hip joint. Total hip replacement (THR) is the most successful application of biomaterials in the short term in order to alleviate joint pain, joint architecture and to increase functional mobility in deceased traumatized joint [3].

Total hip replacement has become a very common operation with over 300 000 operations carried out worldwide every year [4]. Appropriate bone and implant geometry is an important factor for successful total hip arthroplasty. Besides other factors that must be considered are age, weight, bone quality, gender, implant costs. In preparing a THR, an orthopedic surgeon carefully selects an implant from the various type [5].

A typical total hip replacement prosthesis consists of a stem and femoral neck inserted into the top of the femur, a femoral head attached to the femoral neck, an acetabular shell fitted to the hip bone, and an acetabular cup liner fixed to the inside of the acetabular cup and against the femoral head [6]. The design of a hip implant involves many parameters, which include stem length, cross-section, neck length, neck angle, and ball diameter [7].

Numerous studies have been conducted on the use of FEM to determine the strength of hip stem. Bannet used FEM to analyse six hip designs. The results obtained are the stress and displacement of the hip stem. One of the most important factors is cross-section [7]. Mbulelo and Goswami conducted research with FEM to determine the effect on stem interfacial motion. Implants of distinct cross section and stem profile were analyzed [8]. The study also analyzed material which contributed to the stem interfacial motion. The study showed that implant stem cross sectional area may have an effect on the stability and interfacial motion. In addition, Ti 6Al-4V showed the greatest difference in implant interfacial motion as compared to SS316L and CoCr [8]. Kluess conducted a study of the effect of femoral head size on impingement, dislocation, and stress distribution [9]. Galanis N, Manolakos D con-
ducted design of a complete hip joint implant, textural from stem, femoral head and cup, and then used CAE software to analyse and examine of these parts and finally produced inspection of these part for its surface roughness accuracy [10].

The aim of this project was to investigate meshing strategy on mechanical behaviour of hip stem implant design using finite element analysis. In this study, researcher also compared SS 316 L to Ti materials.

2 Materials and Methods

2.1 Design

Many types of hip stem are marketed commercially in Indonesia. Commonly it is imported product. The hip stem design used in this research consists of two components: femoral head dan stem. Femoral head design refers to the study conducted by Tuan P N et al. according ISO 7206-1 as Figure 1 (a) [11]. Commercial hip stem is used as a reference for shape. The shape of the neck and the cross section which were made is illustrated in Figure 1 (b). Smaller neck in the design is unavoidable because there would be contact to the cup of the acetabulum. If possible, even a slight increase in the neck area will certainly have positive implications [12]. In this study rectangular cross sections are used on all types of hip stem.

The length of the intramedullary stem of the femoral prosthesis varies in a large scale from 12 to 18 cm for standard prosthesis [13]. The chosen stem length was 128 mm. In the stem, there are two slot holes. Hip stem design and slot hole is shown in Figure 2.

2.2 Materials

The selected material needs a requirement as implant material. In this study the material chosen was Titanium (Ti-6Al-4V) compared to SS 316 L. Metal biomaterials are extensively used in medical applications due to their high strength and corrosion resistance. In addition, stainless steel, Ti, magnesium and Co-based alloys have superior biomedical properties among other metal biomaterials [14]. Following are the unique properties that made titanium the ultimate choice as a biomaterial in total joint replacements:

- Inert to human body fluids
- Osseointegration capability
- Strong, yet light weight
- Non-magnetic [15]

The material properties of Titanium (Ti-6Al-4V) and SS 316 L are shown in Table 1. The data of mechanical properties of materials was from literature.
Table 1: Mechanical Properties of Materials [16, 17]

| No | Material     | Elastic modulus (GPa) | Ultimate tensile strength (MPa) | Yield Strength (MPa) | Poisson’s ratio | Density (g/cm$^3$) |
|----|--------------|-----------------------|---------------------------------|----------------------|-----------------|-------------------|
| 1  | Ti (Ti-6Al-4V) | 114                   | 900                             | 880                  | 0.32            | 4.4               |
| 2  | SS 316 L     | 200                   | 1,000                           | 314                  | 0.30            | 7.9               |

2.3 Finite Element Analysis

Two different materials hip stem models were analyzed. Stress distribution (von misses) and displacement models were analyzed using ANSYS 16.0. The material properties for all models were assumed to be isotropic, linear elastic and follow the Hooke’s law. Rawal used the number of elements 41000 to analyze the stem for equivalent stress. This study used 9 variations of elements size: 2.5; 2.25; 2; 1.85; 1.75; 1.65; 1.5; 1.35; 1. Method of meshing which was chosen was hex-dom-quad-tri, which is considered based on theory that it is essential to mesh biologic structures with hexahedral elements [2]. In addition, the researcher applied fine mesh to the implant models [15]. The example of the used mesh type is shown in Figure 3.

![Figure 3: The shape of the FE model mesh used.](image)

All models were set to the same boundary conditions. Model was set up with adequate boundary conditions, including fixing of the implant bottom surface along all degrees of freedom, and the load was applied in the appropriate direction relative to the top of the femoral head of the prosthesis [18]. The type of connections contact chosen between the femoral head and neck was bonded. The analysis setting consists of fixed force and force. Fixed boundary condition is set-up along the stem in all orientations as shown in Figure 4. While the magnitude of force was 2,700 N in the femoral head with an angle of 16° C from the vertical axis and 78° C from the horizontal axis. The magnitude of the load component was determined based on literature studies from Pachioga et al. Figure 5 shows the magnitude of the load.

![Figure 4: Fixed force](image)

3 Results and Discussion

Two type materials have been analyzed with FEM to determine the meshing strategy that produces optimum mechanical properties. The results taken from the simulation are stress (von misses), deformation, strain, and Central Processing (CP) time. All the results will be presented in the form of graphs and in the form of two dimensional images. In the selected element size variations, the stress (von misses) for each materials is shown in Figure 6. The graph shows that the stress (von misses) max for Titanium
is 198.93 MPa at 1.85 element size. While the stress (von misses) max for SS 316 L is 188.37 MPa at 1.25 element size.

Deformation generated by the elements size variations is shown in Figure 7. For SS 316 L material, the resulting curve tends to be linear in all element size variations. While the resulting deformation for Ti-6Al-4V material continues to decrease at element size <1.5 and increase at element size >1.65 and linear up to element size 2.5. The resulting smallest deformation is 0.032287 mm at element size 1.5 for Ti and 0.020640 mm at element size 2.5 and 2.25 for SS 316 L.

The relationship between element size and CP time of Ti-6Al-4V and SS316L are shown in Figure 9. The graph shows CP time is decreasing at element size 1.00 to 2.5 for both Ti and SS 316 L. Maximum CP time is generated by element size 1.00 for Ti and SS 316 L. Maximum CP time for Ti is 231,719 seconds and 264,328 for SS 316 L. From the simulation that has been done, the results of stress (von
Figure 10: For element size 1 mm on Ti-6Al-4V (a) Stress (von misses), (b) deformation, (c) strain (von misses) and on SS 316 L 4V (d) Stress (von misses), (e) deformation, (f) strain (von misses)

Figure 11: For element size 2 mm on Ti-6Al-4V (a) Stress (von misses), (b) deformation, (c) strain (von misses) and on SS 316 L 4V (d) Stress (von misses), (e) deformation, (f) strain (von misses)
misses), deformation, strain (von misses) and CP time are obtained. The Figure 10 to 12 show the results of the simulation that has been carried out on meshing size 1mm, 2mm, 2.5mm. In other meshing sizes, the results of contour are almost similar except for the resulting value.

### 4 Conclusion

A finite element analysis has been carried out with variations of the meshing strategy for the two types of implant material. Mesh size variations have an influence on mechanical properties. It is necessary to determine the right mesh size to get the right analysis and optimum CP time. Two types of material have been compared with FEA. From the simulation, SS 316 L material has smaller deformation than Ti material. Whereas CP time decreases in increasing element size for both materials. From this study the recommended mesh size is 2 mm. More variations in mesh size are needed to get more accurate convergent results.

### References

[1] Amirouche F, Solitro GF. Challenges in modeling total knee arthroplasty and total hip replacement. Volume 2. 2011. pp. 18–25.

[2] Shin J, Lee DS, Jeong, Lee HG, and Kim J. 136-701, r, vol. 15, no. 1, pp. 57–65, 2011.

[3] Affatato S. Development of a Novel in Silico Model to Investigate the Influence of Radial Clearance on the Acetabular Cup Contact Pressure in Hip Implants. 2018. pp. 1–11.

[4] Darwish SM. Optimization of Artificial Hip Joint Parameters, no. 3, 2009.

[5] Jun Y, Choi K. Advances in Engineering Software Design of patient-specific hip implants based on the 3D geometry of the human femur. Adv Eng Softw. 2010;41(4):537–47.

[6] Hocking L, Pramanik A, Basak AK, Chattopadhyaya S. Designing and analysis of the femoral neck for an artificial hip joint prosthesis. Elsevier Ltd.; 2019. https://doi.org/10.1016/B978-0-08-102174-3.00002-4.

[7] Bennett D. Materials & Design Finite element analysis of hip stem designs. Volume 29. 2008. pp. 45–60.

[8] Makola MT, and Goswami T. Hip implant stem interface motion, a finite element analysis Hip implant stem interface motion, a finite element analysis Mbulelo T. Makola* Tarun Goswami, no. February 2015, 2011.
Kluess D, Martin H, Mittelmeier W, Schmitz K, Bader R. Influence of femoral head size on impingement, dislocation and stress distribution in total hip replacement. Volume 29. 2007. pp. 465–71.

Galanis N, and Manolakos D. Design, Numerical Analysis and Manufacturing of Artificial Hip Design, Numerical Analysis and Manufacturing, no. June, 2009.

Chi H, City M, Trang N. Design of Femoral Head in Total Hip for Vietnamese Patients. 2016;(June):3–7.

Khouzani MK, Bahrami A, Eslami A. Metallurgical aspects of failure in a broken femoral HIP prosthesis, Eng. Fail. Anal., vol. 90, no. November 2017, pp. 168–178, 2018.

El-shiekh HEF, and Hashmi MSJ. Replacement under Static and Dynamic, no. March, 2002.

Khorasani AM, Goldberg, Doeven MEH, and Littlefair G. Titanium in Biomedical Applications — Properties and Fabrication. RE:view. 2015;5(8):593–619.

Rawal BR, Ribeiro R, Malhotra R, Bhatnagar N. Design and manufacturing of femoral stems for the Indian population. J Manuf Process. 2012;14(3):216–23.

Sabatini AL, Goswami T. Materials & Design Hip implants VII: Finite element analysis and optimization of cross-sections, vol. 29, pp. 1438–1446, 2008.

Kocich R, Lowe TC, Kunc L. Progress in Materials Science Advances in metals and alloys for joint replacement. Volume 88. 2017. pp. 232–80.

Colic K, Sedmak A, Grbovic A, Tatic U, Sedmak S, Djordjevic B. Finite Element Modeling of Hip Implant Static Loading. Procedia Eng. 2016;149(June):257–62.

Pacioga A, Palade DD, and Scomșa. Volume 73. Computational Simulation of Bone-Personalized Hip Prosthesis Assembly; 2011.