The Sensitivity Analysis in Topology Optimization of Hip Stem Prosthesis Using Finite Element Method

SYIFAUL Huzni¹, M. ILHAM Tanamas¹, SYARIZAL Fonna¹ and A.K. ARIFFIN²

¹Dept. of Mechanical and Industrial Engineering, Faculty of Engineering, Universitas Syiah Kuala, Banda Aceh, Indonesia
²Centre for Integrated Design for Advanced Mechanical Systems (PRISMA), Universiti Kebangsaan Malaysia

E-mail: syifaul@unsyiah.ac.id

Abstract. This paper discusses the results of the parameters employed sensitivity analysis in the optimization of hip stem prosthesis geometry using topology optimization techniques based on finite element methods. The study was carried out to examine the relationship of finite element parameters, such as mesh size and stress distribution, and topology optimization parameters, such as percent to retain, to the final geometry of the prosthesis stem. Topology optimization was performed using ANSYS software for hip stem prosthesis, a type of Anatomic Medullary Locking, made from titanium alloy. The results obtained from this study indicate that the selection of finite element parameters and topology optimization parameters will have a significant effect on the shape and geometry of the hip stem. The results of this analysis can be the basis for the development of hip stem shape and geometry through topology optimization techniques.

1. Introduction

Topology optimization based on the finite element analysis is one of the optimization techniques that is often employed in the process of shape optimization. This technique has been implemented in various fields of science, such as automotive, structural engineering, aerospace, and biomechanics, to produce a more optimal design [1]. Topology optimization works by taking a solid block of material in any shape and removes material from it to minimize or maximize an optimization objective such as mass, displacement, or stress.

Numerically, the process of analysis and optimization involves several parameters that make up the model used in the analysis. Variations in the value of the parameters used may cause differences in the results obtained. To reduce the effect of uncertainty on the optimization results that will be achieved when optimization is carried out, it is necessary to analyze the sensitivity of the parameters used in finite element analysis and also the parameters used in topology optimization. Therefore, in this paper, we will discuss the results of investigations on the two parameters employed in the hip-stem-prosthesis optimization process, i.e., mesh size and percent of retain
2. Geometry and Material

The dimensions of the prosthesis used in this study refer to those developed by previous researchers, namely based on anthropometric data of the human body of Indonesian people aged between 22-50 years old. Previous researcher has conducted an analysis related to the safety level of hip stem prosthesis, Anatomic Medullary Locking type, using geometry adjusted for Indonesian people's body posture, with a stem length of 140 mm, a ball head diameter of 38 mm and an extrusion thickness of 15 mm [2]. Detailed dimensions of the model used in this study are shown in Fig. 1. The boundary conditions given to the prosthesis for finite element analysis are shown in Fig. 2, with the force component for standing conditions of $F_x = 0\% \text{ BW}$; $F_y = -100\% \text{ BW}$; $F_z = 0\% \text{ BW}$, where the body weight (BW) selected is 70 kg [3,4]. Material properties used in finite element analysis for hip stem prosthesis refer to Ti-6Al-4V titanium alloy material, with a poisson ratio ($\nu$) = 0.342, modulus of elasticity ($E$) = 113.8 GPa [5].

![Figure 1. Geometry of prosthesis](image1)

![Figure 2. Boundary Condition](image2)

3. Control parameters

**Finite element Analysis.** The type of mesh employed in this study is tetrahedrons. Three variations of the mesh size used are fine, medium and coarse, according to the default conditions in ANSYS software. When examined in more detail, the average mesh size for fine is 3 mm, medium is 5 mm and coarse is 10 mm. The arrangement of the mesh in the hip stem prosthesis based on its size can be seen in Fig. 3.
Topology Optimization. In the topology optimization procedure, one important control parameter is percent-of-retained (POR). POR is used as a determinant of the area that can be optimized with the topology method. In this study, three POR values were chosen namely 40%, 60% and 80%, with minimizing compliance as an objective. Determination of the area removed is based on the distribution of stress magnitude from the finite element analysis results.

4. Result and Discussion

**Correlation between mesh size and stress distribution.** The results of finite element analysis to study the effect of mesh size on the stress distribution of the prosthesis used for standing conditions are shown in Fig. 4. The highest concentration of von-mises stress is in the neck of the hip stem prosthesis. The maximum value of von-mises stress for mesh coarse size is 41,136 MPa, medium 47,424 MPa, and fine 50,609 MPa. The differences occur due to the differences in the mesh size used in finite element analysis, as can be seen in Fig. 4 and Fig. 5.

![Figure 3. Meshing (a) Coarse, (b) Medium, (c) Fine](image)

![Figure 4. Von Misses stress distribution of pre optimization prosthesis a) Coarse, (b) Medium, (c) Fine](image)
The relationship between mesh size and optimization results. The simulation results that have been carried out show that mesh size influences the final form of optimization results with topology techniques for hip stem prosthesis. The results of topology optimization show that removing only occurs in part of the predetermined design region. The process of removing that occurs is based on the level of stress that occurs. Parts of the prosthesis model that experience minimum stress will be removed element by element [6]. The size of the removed model depends on what percentage to retain (POR), and the size of the mesh used also has an effect on the removal process as shown in Fig. 6.

Stress Analysis of the optimized model. The simulation results show that there are differences in stress distribution between models that have been optimized using topology optimization techniques. Nevertheless, the stress distribution on the prosthesis shows an even distribution, not only focusing on a certain area. The maximum von-Mises stress value for the prosthesis model with 80% to retain is 87,836 MPa, 60% to retain is 76,257 MPa, and 40% to retain is 126.9 MPa.

Prosthesis with 40% to retain shows the highest maximum von-mises stress magnitude, compared to the model with 60% or 80% to retain as shown in Fig. 7. In this case, the smaller percent to retain is used, the von Mises stress distribution will be more evenly distributed, and no longer centered on the neck of the hip stem prosthesis as found in prosthesis that has not been optimized. Even so, the maximum value of von Misses stress occurred is still below the value of material yield, so it can still be considered safe to apply.
Figure 6. Mesh vs. Retain Percentage (a) 80%, (b) 60%, (c) 40%
Summary

The results of sensitivity analysis that have been carried out in this study show that the mesh size, which is a finite element analysis parameter, and the retain percentage, which is a parameter of topology optimization, influences the final shape and geometry of the optimization results on the hip stem prosthesis. Fine mesh gives the analysis results of a more sensitive Fine mesh that provides the results of stress analysis with better sensitivity, this can be seen from the magnitude of stress that is greater than other mesh sizes. Mesh size also influences on the shape and final geometry of the optimization of the hip stem which is done by topology techniques.

Acknowledgement.

The research was supported by Penelitian Lektor Sumber Dana PNBP Universitas Syiah Kuala 2019 with contract number of 224/UN11.2/PP/PNBP/SP3/2019.

Figure 7. Von Misses stress distribution of post optimization prosthesis on fine mesh
(a) 80%, (b) 60%, (c) 40%
References

[1] Martin P. Bendsøe & Ole Sigmund, *Topology Optimization: Theory, Methods, and Applications*, Springer-Verlag, New York, 2004

[2] Syifaul Huzni, Fitri Handayani, Syarizal Fonna & M. Ridha, *Analisis Hubungan Dimensi Stem dan Ballhead Terhadap Faktor Keamanan Fatik Hip Stem Prosthesis dengan Menggunakan Metode Elemen Hingga*, Prosiding Seminar Nasional Tahunan Teknik Mesin XIV, Program Studi Teknik Mesin Universitas Lambung Mangkurat (2015)

[3] Syifaul Huzni, Rizky S, Syarizal Fonna, Sulaiman Thalib, M. Ridha, A.K Arifin, *Analisa Distribusi Tegangan pada Hip Stem Prosthesis dengan Menggunakan Metode Elemen Hingga*, Prosiding Seminar Nasional Rekayasa (SNTR) III Tahun 2016, Politeknik Aceh Selatan (2016).

[4] *Loading of Orthopedic Implants*, information on www.otrhoload.com/database/

[5] *Titanium Ti-6Al-4V (Grade 5) Annealed*, information on http://asm.matweb.com/search/SpecificMaterial.asp?bassnum=mtp641

[6] Picelli, R., Townsed, S., et al., *Stress-based Shape and Topology Optimization with the Level Set Method*, J Computer Methods in Applied Mechanics and Engineering. (2017) pp. 1-23.