

Application of Topology Optimization in Design of Stem Profile in Hip Implants using Finite Element Method

Mohammad Reza Niroomand *

Department of Mechanical Engineering,
Payame Noor University, Iran
E-mail: niroomand@pnu.ac.ir
*Corresponding author

Farzad Boroomand

Department of Mechanical Engineering,
Najafabad Branch, Islamic Azad University, Isfahan, Iran
E-mail: farzad.b@smc.iaun.ac.ir

Received: 28 January 2018, Revised: 14 March 2018, Accepted: 10 May 2016

Abstract: Appropriate design of stem shape is an important factor in total hip replacement. It affects the attachment of the implant to the bone and the stability of the implant. Using topology optimization, this study has been an attempt to propose an optimized model of the stem profile of the hip implants. In this regard, a three-dimensional finite element model of the implant has been combined with a stiffness-based topology optimization algorithm to reduce the relative motion between the implant and the bone. The objective function in the optimization problem is the compliance of the structure which should be minimized. Also, a constraint on usable volume is applied to the structure. Then smoothing process has been done on the optimal model to prevent its geometric complexities. Results show that the final model has a smaller weight, less displacement, and more uniform stress distribution. In addition, using conventional production methods, this model can be easily produced.

Keywords: Finite Element, Hip Implant, Stem Profile Design, Stress Analysis, Topology Optimization

Reference: Mohammad Reza Niroomand and Farzad Boroomand, "Application of Topology Optimization in Design of Stem Profile in Hip Implants Using Finite Element Method Title", *Int J of Advanced Design and Manufacturing Technology*, Vol. 11/No. 2, 2018, pp. 95-101.

Biographical notes: **M. R. Niroomand** received his BSc, MSc and PhD from Department of Mechanical Engineering, Isfahan University of Technology (IUT), Isfahan, Iran in 2004, 2006 and 2012, respectively. His current research focuses on the biomechanics, design of experiments, optimization, welding, and metal forming. **F. Boroomand** received his BSc from Department of Mechanical Engineering, Isfahan University of Technology (IUT), Isfahan, Iran in 2011 and his MSc from Department of Mechanical Engineering, Islamic Azad University of Najafabad, Isfahan, Iran in 2014 and is currently a PhD candidate in Department of Mechanical Engineering University of Birjand, Birjand, Iran.

1 INTRODUCTION

Every year a large number of hip replacement surgeries are performed all around the world. This considerable figure shows the necessity for designing and producing more capable implants. One of the problems of the hip implants is the looseness of them after the surgery. This problem is usually due to the inadequate Osseo integration between the implant and bone because of the factors such as inappropriate stress distribution in the implant. Several areas are effective in the design of the hip implants. The shape of the stem is an important issue to optimize the longevity and functionality of the hip replacement. Some researches have investigated the effect of geometrical factors in the design of hip implants. Kayabasi et al. analyzed four different stem shapes in order to achieve an optimal design of hip implants [1]. They examined different geometries with varying curvatures under static and dynamic loading using finite element method.

Bennett et al. using finite element method compared the implants with circular and square cross-section [2]. The cross-section that comprised a circle in the medial end and a square at the lateral end was found to provide suitable design characteristics. Sabatini et al. analyzed several kinds of hip implants with different designs [3]. They found that hip stem profile with the trapezoid cross-section showed the lowest von Mises stress in most of the implant regions. Wang proposed a method for designing customized implants for the specific patient [4]. He used material-property analysis and fuzzy matter-element analysis to select the best structure and materials for the customized prosthesis.

Nowadays, many studies have been conducted in the field of topology and shape optimization to redesign implants and prostheses. Boyle et al., using topology optimization, could find the optimized shape of the hip implant [5]. They tried to pay attention to the implant behavior after surgery and during the bone healing. To this end, they performed the optimization process on a micro-level finite element model. Fraldi et al. used topology optimization in order to design hip implants [6]. By optimized design, they aimed to minimize the risk of implant fracture. In this regard, they tried to reduce stress concentration at the contact surfaces between the implant and the bone. Virulsri et al., using multi-objective shape optimization designed a hip implant by finite element method [7]. Changing different parameters related to the implant shape, they tried to perform an optimal design suitable for the physical requirements of Thai patients. Katoozian et al. tried to optimize the shape of the hip implant [8]. They tried to reduce relative motion between the implant and the bone. The main concept was to reach prostheses with a more rectangular cross-section. Tanino et al., combining the methods of sensitivity analysis and shape

optimization, tried to offer an optimal design of the hip implant [9]. Using sensitivity analysis, at first, they identified the parameters that had a significant effect on the performance of the implant and, then, using shape optimization, obtained the optimal values of the parameters.

In previous research, different methods have been used to improve the stability of the implant. One of these solutions is the use of topology optimization to create some cavities within the implant. In this case, the bone grows and it moves into the cavities, but the implant strength is usually reduced and Osseo integration between the implant and the bone takes longer. Through using topology optimization, the current study aims at providing an optimal profile of the stem of the hip prostheses. In the first part of this study, finite element model of the implant was developed. Then, topology optimization technique was employed to generate an optimized profile of the stem according to femoral loads and constraints. After optimization, the smoothing process is done to reduce geometric complexity.

2 FINITE ELEMENT ANALYSIS

Finite element method is widely used for modeling biomechanical processes and structures [10-11]. It is also a virtual alternative for clinical techniques and experiments [12]. The finite element model can then be used for topology optimization at next steps. ABAQUS is one of the most powerful finite element softwares. In this research, modeling and optimization processes are done in this software.

2. 1. Modeling

2. 1. 1. Geometrical modeling

The performed model includes a femur and an implant. As previously mentioned, hip stem profile with the trapezoid cross-section showed the lowest von Mises stress in most of the implant regions. So, a hip implant with a straight stem and trapezoid cross-section is considered. There is a cornered shoulder on the lateral side as can be seen in "Fig. 1".

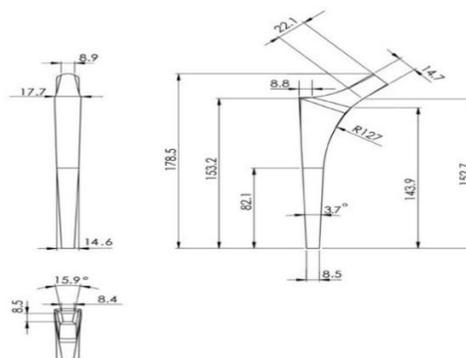


Fig. 1 Hip profile.

Three-dimensional models of the implant, femur and their assembly are shown in “Fig. 2”.

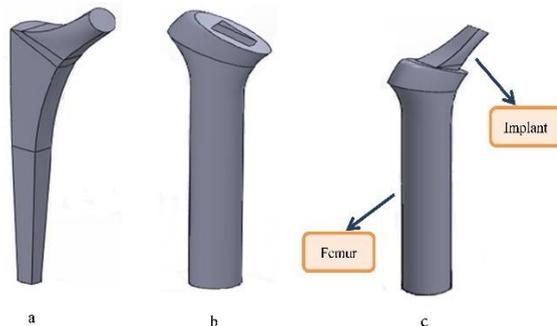


Fig. 2 Three-dimensional model: (a): implant, (b): femur and (c): assemble.

2. 1. 2. Material properties

Despite the wide variety of biomaterials, titanium is still used largely in the orthopedic applications. This is mainly due to the appropriate tensile strength, low weight, and excellent corrosion resistance as well as the spontaneous creation of a passive oxide layer [13]. Owing to good properties of titanium alloys, Ti-6Al-4V has been used for implant material. Behaviors of the materials used in the finite element model were considered as isotropic linear elastic [3]. Mechanical properties of these materials can be seen in “Table 1”.

Table 1 Mechanical properties of the materials used in the finite element model

| Material | Elastic Modulus (GPa) | Ultimate Strength (MPa) | Poisson’s Ratio | Density (g/cm ³) |
|---------------|-----------------------|-------------------------|-----------------|------------------------------|
| Ti-6Al-4V | 114 | 900 | 0.32 | 4.4 |
| Cortical bone | 20 | 130 | 0.30 | 2.0 |

2. 1. 3. Loading conditions

The loading includes a 3000 N force that has been applied normal to the implant neck [3]. The negligible force of the tendons has been ignored and only the forces applied to the femur head have been considered. In order to prevent stress concentration, loading has been applied as a distributed load. The applied load is usually considered to be 0.5 to 4 times greater than the person’s weight. In order to include the effect of dynamic loads in static modeling, the amount of applied load is usually considered to be 4 times greater than the person’s weight [3]. Assuming that the weight of the person is 75 kg, a 3000 N force is applied.

2. 1. 4. Boundary conditions

To apply boundary conditions, all degrees of freedom of the nodes connected to the distal femur were fixed. In addition, the femur was constrained at the end of the proximal and near the greater trochanter. Figure 3 shows

the boundary conditions of the finite element model. At the contact surfaces, frictional conditions with a coefficient of friction of 0.3 were applied.

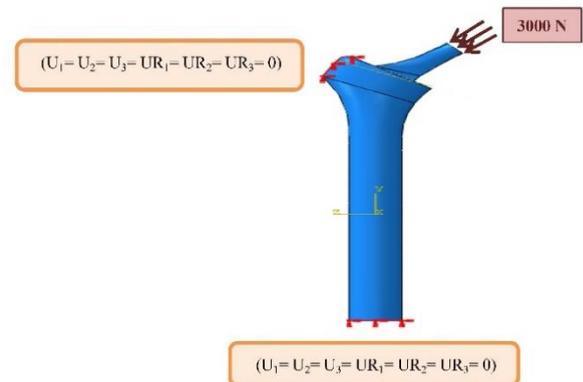


Fig. 3 Loading and boundary conditions in the finite element model.

2. 1. 5. Meshing

The obtained model, using 5909 elements, was meshed. The types of used elements included 10-node quadratic tetrahedral element. In order to confirm the results of the analysis, mesh convergence should be evaluated. So, various analyses were performed with a different number of elements. Finally, the model meshed in such a way that the results were independent of the number of elements.

2. 2. Simulation Results

In order to evaluate the performance of the hip implant, von Mises stress was assessed at four important regions of the implant stem. These four regions include: 1) medial of proximal, 2) medial of distal, 3) lateral of distal, and 4) lateral of proximal. The location of these four regions is shown in “Fig. 4”.

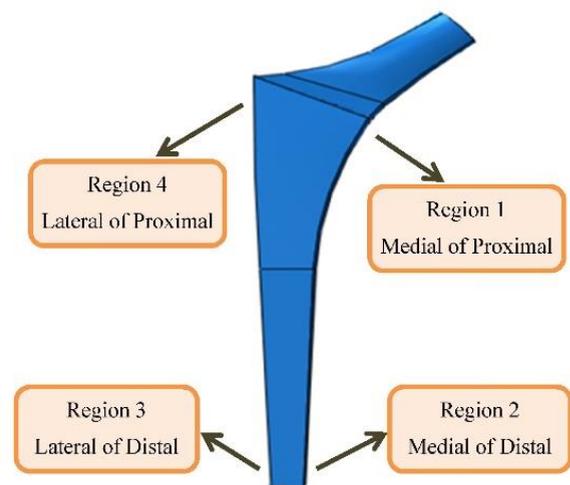


Fig. 4 The location of the four investigated regions.

Von Mises stress distribution at the stem together with the values of stress in the four mentioned regions has been shown in “Fig. 5”. Stress values reported in these four regions are an average of the stress in the nodes of these regions.

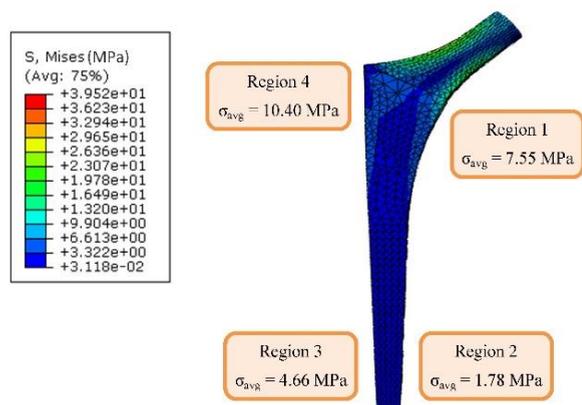


Fig. 5 Von Mises stress distribution at the stem.

Stress values are compared with the result of Sabatini et al. [3] in “Table 2”. Comparing these values, it can be seen that the results are in a good agreement.

Table 2 Von Mises stress values

| Average of Von Mises Stress (MPa) | | | | |
|-----------------------------------|----------|----------|----------|----------|
| | Region 1 | Region 2 | Region 3 | Region 4 |
| Present model | 7.55 | 1.78 | 4.66 | 10.40 |
| Sabatini et al. [3] | 7.53 | 1.63 | 6.17 | 9.80 |

As shown in figure 5, stress distribution is non-uniform in the stem. The stress in the proximal end is more than the stress in the distal end. The stress in the middle part is much less than the stress in the proximal and distal end. The amount of stress has been increased in the distal and proximal parts, which is due to the stress concentration. The stress in the medial and lateral parts has also distributed non-uniformly. The amount of stress is low on the medial side, while it is high on the lateral side.

In general, according to the stem shape, mass distribution in the implant stem is in such a way that stress has not been distributed uniformly. Therefore, in some areas of the stem, where there is no high stress, the material can be removed. In other words, by changing the implant design, mass can be distributed in accordance with the stress distribution. Based on the performed modeling and using topology optimization technique, the aim of this study is to propose an optimal profile of the stem with more efficiency.

3 TOPOLOGY OPTIMIZATION

Topology optimization is a mathematical approach that determines the optimal material distribution for a part [14]. In this approach, the basic shape of the part is given to the algorithm and some regions are removed from the original part. This study is an attempt to use topology optimization technique in designing the stem profile in the hip implants. The optimal design has been performed to achieve an implant with a smaller volume, less displacement, and a more uniform stress distribution.

3.1. Topology Optimization Algorithm

Topology optimization is an approach to optimize the material layout. It generates new conceptual designs by modifying material distributions [14]. It has been widely used for designing structures and materials for desirable performance and physical properties [15]. After completing the finite element model and making sure about the model results, it can be used for optimization. The goal of topological optimization is to find the best use of material for a body to maximize its performance subject to given constraints.

Solid Isotropic Microstructure with Penalization (SIMP) algorithm which is implemented in ABAQUS was used for optimization. Starting from the base idea offered by the homogenization method, SIMP algorithm adjusts the density and stiffness of the design variables while trying to minimize the objective function and satisfy the constraints.

In the stiffness-based topology optimization algorithms, similar to what is used in this research, the strain energy is used as objective function to maximize the global stiffness [16]. In a finite element model, the compliance of a structure is defined as the sum of the strain energy of all the elements, $u^T Ku$ for linear models, where u is the displacement vector and K is the global stiffness matrix. Compliance or flexibility is the inverse of the stiffness and minimizing the compliance in the finite element model is equivalent to maximizing the global stiffness of the model.

In an optimization problem, the design variables represent the parameters to be changed during the optimization. In topology optimization, the densities of the elements in the design area are changed during each iteration of the optimization and couples the stiffness of each element with the density. SIMP is a “soft-kill” algorithm. In this method, a variable called pseudo-density is defined. This variable, x_j , which varies between zero and one, specifies the deletion or existence of the j^{th} element.

$$x_j = \frac{\rho_j}{\rho_0} \quad (1)$$

Where ρ_j is the density of the j^{th} element, ρ_0 is the density of the base material and x_j is the pseudo-density of the j^{th} element. As can be seen in table 1, the value of ρ_0 is 4.4 for Ti-6Al-4V.

Design variables in the optimization problem are the pseudo-density of the elements. These variables are gathered in a vector and represent the vector of design variables. In this research, the number of elements that determines the number of design variables is 5909.

As mentioned before, the objective function is the compliance of the structure which should be minimized. A constraint on usable volume is applied to the structure. If the volume constraint does not apply to the optimization problem, optimization algorithm simply fills the entire design area with material. So the optimization problem can be expressed as:

$$\text{Objective function: } c(x) = u^T K u = \sum_{j=1}^n u_j^T k_j(x_j) u_j \quad (2)$$

$$\text{Constraints: } \sum_{j=1}^n x_j v_j \leq V_0, \quad 0 \leq x_j \leq 1, \quad j = 1, 2, 3, \dots, n \quad (3)$$

Where $c(x)$ is the compliance of the finite element model, V_0 is the initial volume which is 28703.6 mm³ in this model, and n is the number of the elements.

3. 2. Finite Element Topology Optimization

After completing the finite element model and making sure about the model results, it is used for topology optimization. Optimization is an iterative process that updates the design variables, executes an analysis of the modified model, and reviews the results to determine if an optimized solution has been reached. Each optimization iteration is called a design cycle.

“Fig. 6” indicates how the optimization algorithm tries to minimize objective function while satisfying the constraints. It can be used as a diagnostic tool to view the progression of the optimization after each design cycle and to determine if the optimization is converging on a solution.

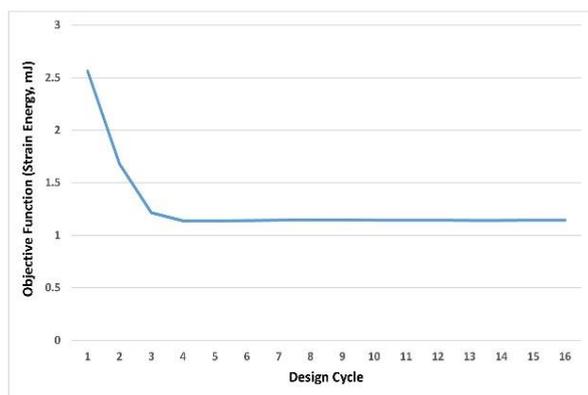


Fig. 6 Objective function during optimization.

As can be seen, optimization process was performed over 16 cycles. “Fig.7” shows the optimized model. The volume of the optimal model has decreased up to 16% compared to the initial model.

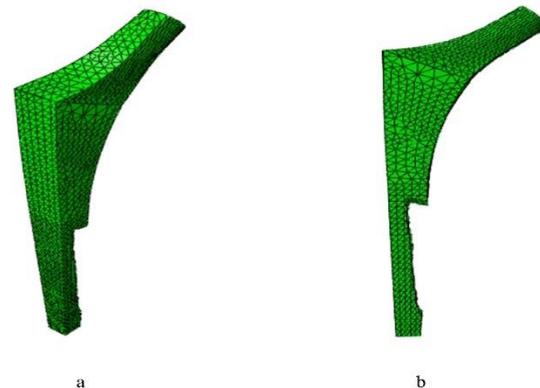


Fig. 7 The model resulted from topology optimization: (a): isometric view and (b): two-dimensional view.

In the optimization process, it is expected that the mass is eliminated from the fewer stress areas. In figure 5, it was shown that stress in the distal part is less than the proximal part. Moreover, stress on the medial side is less than the lateral side. Evaluation of the optimal model shows that the removed areas have been started from the medial side of the distal area and continued to the middle area of the stem. In this regard, it is expected that reducing the volume of the part does not reduce its strength.

3. 3. Smoothing Process

Topology optimized model should be such that the implant can be produced easily. Therefore, after topology optimization, smoothing process has been carried on the optimal model. The modified model is so that can be produced by the conventional processes and is shown in “Fig. 8”. Details of the optimized model which have changed in comparison with the original model have been dimensioned in figure 8 (part b). Other dimensions of the optimized model are similar to the original model.

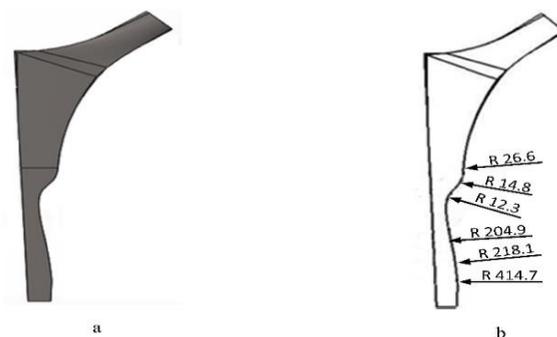


Fig. 8 Optimized model: (a): a view of the modified implant and (b): dimensions.

It is worth noting that the final optimized model, which is achieved by smoothing process, is a little more complex than the original model but can be produced by the conventional methods.

4 DISCUSSION

The final model was re-analyzed by the finite element method. “Fig. 9” shows von Mises stress distribution at the stem before and after the optimization.

As can be seen, the values of von Mises stress have become close to each other in the four mentioned regions. In other words, stress has been distributed more uniformly in the optimized model. Therefore, volume reduction not only has not increased stress in the implant but also has decreased the maximum stress value. “Fig. 10” shows displacement distribution at the stem before and after the optimization.

Comparison of the displacement values in both models shows that displacement of all four regions has been decreased in the optimal model. Moreover, the maximum displacement in the implant has decreased compared with the initial model. Thus, reducing the volume of the implant not only has not increased its displacement but also has decreased it in the optimal model. Reduced displacement prevents the loosening of the implant and increases its stability inside the bone. The process to reach the final model is shown in “Fig. 11”.

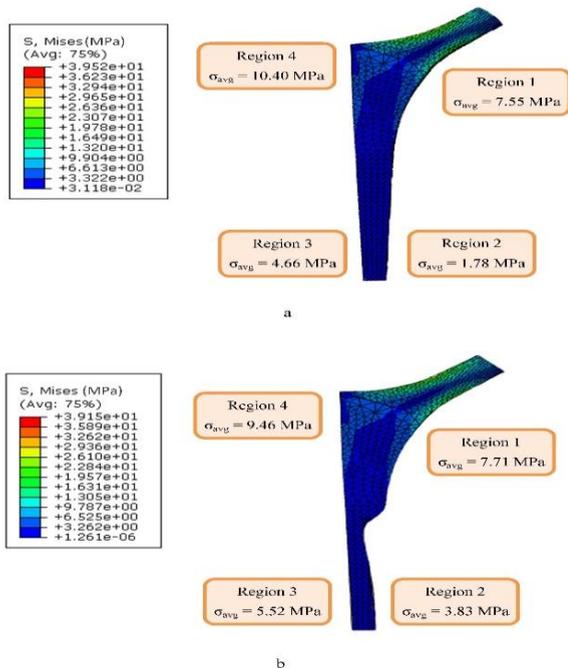


Fig. 9 Von Mises stress distribution at the stem: (a): before optimization and (b): after optimization.

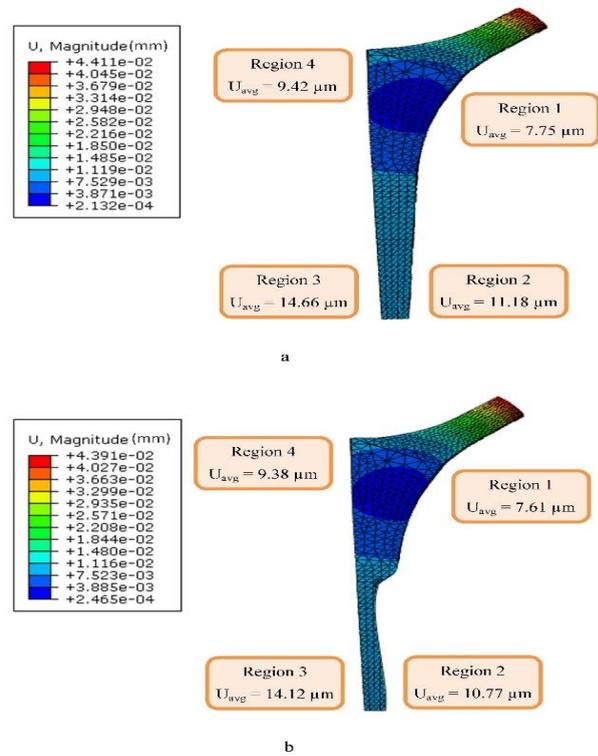


Fig. 10 The amount of total displacement in the stem: (a): before optimization and (b): after optimization.

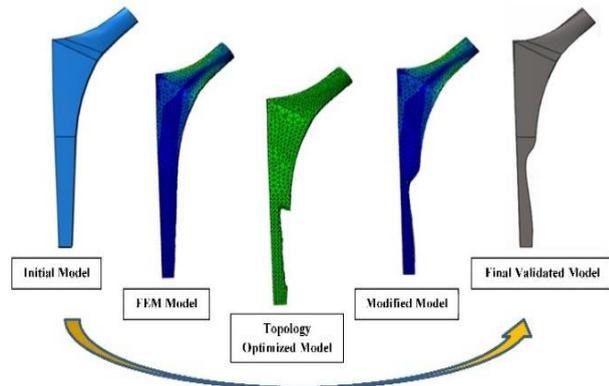


Fig. 11 The process of designing the optimal implant.

As can be seen, the final model, in addition to better performance, does not have much complexity. Selecting strain energy as the objective function led to more strength and less displacement in the implant. On the other hand, in decreasing the volume, those areas were removed which endure less stress. Thus, stress values became closer in the remaining areas. In other words, by choosing strain energy as the objective function, the process of volume reduction led to a more uniform stress distribution.

5 CONCLUSION

An optimal profile of the stem of hip implant was offered in this study. To this end, at first, a finite element model of the implant's initial design was prepared. Then, using topology optimization, an optimal design of the implant was tried to be achieved. In the optimization process, strain energy was considered as the objective function and volume reduction as the constraint function. In this process, volume reduction continued until the part strength had not reduced. After doing topology optimization, the model was modified in order to reduce its complexity and facilitate the production process. The results showed that the optimal model is up to 16% lighter than the original model. Moreover, stress was distributed more uniformly in the optimal model and displacement was reduced in the important regions.

REFERENCES

- [1] Kayabasi, O., Erzincanli, F., Finite Element Modelling and Analysis of a New Cemented Hip Prosthesis, *Advances in Engineering Software*, Vol. 37, No. 7, 2006, pp. 477-483.
- [2] Bennett, D., Goswami, T., Finite Element Analysis of Hip Stem Designs, *Materials & design*, Vol. 29, No. 1, 2008, pp. 45-60.
- [3] Sabatini, A., Goswami, T., Hip Implants VII: Finite Element Analysis and Optimization of Cross-Sections, *Materials & Design*, Vol. 29, No. 7, 2008, pp. 1438-1446.
- [4] Wang, M., Optimization of Femoral Prosthesis Based on Comprehensive Evaluation of Structure and Material Properties, *Journal of Mechanics in Medicine and Biology*, Vol. 16, No. 8, 2016, pp. 1640013.
- [5] Boyle, C., Kim, I. Y., Comparison of Different Hip Prosthesis Shapes Considering Micro-Level Bone Remodeling and Stress-Shielding Criteria Using Three-Dimensional Design Space Topology Optimization, *Journal of Biomechanics*, Vol. 44, No. 9, 2011, pp. 1722-1728.
- [6] Fraldi, M., Esposito, L., Perrella, G., Cutolo, A. and Cowin, S., Topological Optimization in Hip Prosthesis Design, *Biomechanics and Modeling in Mechanobiology*, Vol. 9, No. 4, 2010, pp. 389-402.
- [7] Virulsri, C., Tangpornprasert, P. and Romtrairat, P., Femoral Hip Prosthesis Design for Thais Using Multi-Objective Shape Optimization, *Materials and Design*, Vol. 68, 2015, pp. 1-7.
- [8] Katoozian, H., Davy, D. T., Three-Dimensional Shape Optimization of Femoral Component of Hip Prostheses with Nonlinear Frictional Interface, *Journal of Mechanics in Medicine and Biology*, Vol. 4, No. 2, 2004, pp. 161-172.
- [9] Tanino, H., Higa, H. I. M., Omizu, N., Nishimura, I., Matsuda, K., Mitamura, Y., and Matsuno, T., Three-Dimensional Computer-Aided Design Based Design Sensitivity Analysis and Shape Optimization of the Stem Using Adaptive P-Method, *Journal of Biomechanics*, Vol. 39, No. 10, 2006, pp. 1948-1953.
- [10] Sazesh, S., Ghassemi, A., Ebrahimi, R., and Khodaei, M., Experimental and Numerical Analysis of Titanium/HA FGM for Dental Implantation, *International Journal of Advanced Design and Manufacturing Technology*, Vol. 10, No. 1, 2017, pp. 57-74.
- [11] Rahmati, S., Kheirollahi, H., and Azari, A., Design and Analysis of a New Dental Implant using Finite Element Method, *International Journal of Advanced Design and Manufacturing Technology*, Vol. 6, No. 3, 2013, pp. 23-32.
- [12] Munckhof, S. V. D., Sadri Nikooyan, A., and Zadpoor, A. A., Assessment of Osteoporotic Femoral Fracture Risk: Finite Element Method as a Potential Replacement for Current Clinical Techniques, *Journal of Mechanics in Medicine and Biology*, Vol. 15, No. 3, 2015, pp. 1530003.
- [13] Jagtap, K. A., Pawade, R. S., and Giradkar, K. V., Investigations on Surface Integrity and Electrochemical Behavior of Machined Co-Cr-Mo Bio-implant Alloy, *International Journal of Advanced Design and Manufacturing Technology*, Vol. 9, No. 4, 2016, pp. 51-58.
- [14] Xiaojian, W., Shanqing, X., Shiwei, Z., Wei, X., Martin, L., Peter, C., Qian, M., Milan, B., and Yi, M. X., Topological Design and Additive Manufacturing of Porous Metals for Bone Scaffolds and Orthopedic Implants: A Review, *Biomaterials*, Vol. 83, 2016, pp. 127-141.
- [15] Shiwei, Z., Wei, L., Guangyong, and Qing, L., A Level-Set Procedure for the Design of Electromagnetic Metamaterials, *Optics Express*, Vol. 18, No. 7, 2010, pp. 6693-6702.
- [16] Chih Ling, C., Chen Sheng, C., Chang Hung, H., and Hsu, H., Finite Element Analysis of the Dental Implant Using a Topology Optimization Method, *Medical Engineering and Physics*, Vol. 34, No. 7, 2012, pp. 999-1008.