Total Hip Replacement Implant Designing and its Computational Analysis using ANSYS

Thrinayan Moorthy[1,*], Shubhanvit Mishra[2,*], Mohamed Hathim B[3], Palla Deshik Yadav[4], Bhanu Nirosha Yalamandala[1], Thi My Hue Huynh[1]

[1] Department of Biomedical Engineering and Environmental Science, National Tsing Hua University, Hsinchu, Taiwan
[2] Institute of Nanoengineering and Microsystem, National Tsing Hua University, Hsinchu, Taiwan
[3]Department of Biomedical Engineering, SRM Institute of Science and Technology, Kattankulathur, India
[4] Department of Medicine, Gayatri Vidya Parishad Institute of Health Care and Medical Technology, Visakhapatnam, India

***

Abstract - Total hip replacement (THR) is an effective procedure which relieves pain and helps regain the functionality of degenerated femur bones, which might have occurred due to osteosarcoma, bone fracture, revision arthroplasty, or accidents. There are multiple factors which are responsible for the success of an implant which includes implant design, mechanical stability, osseointegration, and a patient's immunological responses, etc. But the most important factor is design of the implant and its mechanical stability. The mechanical stability of an implant determines its durability and life. The implant design determines the stability as it plays a significant role in appropriate fixation and osseointegration. We have used different models and experimental analysis to determine the mechanical characteristics of the total hip implant for different individuals, developing isotropous and aeolotropic models oriented to determine the mechanical activity of the total hip implant. These models give us important information about the resistance of the hip implant, thereby allowing us to predict its functionality in real time. Images from CT scans of healthy individuals have been utilized to analyze the morphology. CT images are extracted using mimics software and were converted to STL file and was used for obtaining the dimensions of the femoral bone, and these dimensions were used to design a patient-specific hip implant. It was designed using Auto-CAD software. FEA method was used to test the mechanical strength of different materials. The dimensions, mass, and volume of the femur bone as well as the angle of the femoral head vary from person to person thereby making it necessary to design a patient specific implant to suit personal need and requirements.

***Key Words*: Total hip replacement, Finite Element Analysis, CAD, Mimics, ANSYS, Patient Specific Implant**

1. INTRODUCTION

To this day, hip replacements have been a very complex and arduous process. The implants used in this particular procedure are often not compatible with the patients and cause discomfort during their day-to-day activities. Our aim is to create a patient-specific hip implant which suits the specific requirements of effected individual. A hip replacement implant is used when a person is suffering from osteosarcoma, femoral neck fractures, and revision arthroplasty [1]. Revision arthroplasty is performed due to inappropriate prosthesis design, incompatible materials and inappropriate surgical and fixation methods [1,2]. A hip joint fracture may occur due to various reasons such as osteoporosis (low bone density), accidents, falling from a height, and sometimes due to diseases such as arthritis [1]. People affected by hip fractures would not be able to lead a normal and healthy life due to severe pain and immunological responses. Thus, in order to lead a healthy and normal lifestyle, the patient has to undergo an arthroplasty or hip replacement surgery. Implant design and structural strength plays a major role in the success of any implant. Improper design and uneven distribution of stress over implant generally leads loosening of the implant [3]. Hence design is an important aspect of any implant because another major problem faced is the damage of bone during surgery[4]. The volume and density of bone present also determines the success and stability of implant [5,6].

In this procedure, the proximal part of the patient's femoral bone and cartilage (the diseased or affected tissue) is surgically replaced with an implant [7]. The affected individuals face inaccurate dimensions in the implant, which causes difficulties in their day to day life. The present-day implants use a common material for every individual, but a different material is used depending on the size, shape, and weight of the individual. The size of the femur bone varies for each individual.

If healthy, a human or animal's bones will have the ability to change their vulnerability towards different amounts of loads which are placed on them. When the amount of load increases the bones of the human or animal will alter them in such a way that it's strength also increases to handle the load and vice versa [8]. Initially, there is an alteration in the structure of the trabeculae bone, as well as a modification in the external cortical bone portion, this causes the entire bone to increase in its thickness. Similarly, if the bone is not loaded as much or given as much load, there will be decrease in the strength and thickness of the bone this phenomenon is stress-shielding [9]. This can lead to a decrease in bone density, which is also known as osteopenia. Although the strength of bones is altered based on the load, pressure, and mechanical properties which they are put under due to their dynamic nature, they are also prone to alterations based on changes in their hormones and metabolic states. The process of bone homeostasis is depicted in situations where the bone's density and strength decrease when it is not being influenced by and loads, or found in a 'weightless' surrounding. Similarly, after an individual experience an injury which does not allow them to use their bones, they find that their bones become weaker than before, due to the lack of load placed on it. Hence it is very essential for the implant to mimic the properties of bone in order to withstand the adequate load which would in-turn prevent the loosening of implant and prevent revision arthroplasty [10].

A patient-specific hip implant was designed according to the requirements and needs of the patient. We measured the dimensions of the femur of the affected individual using a CT scanner. The CT data was extracted using Mimics to obtain a 3D model of the femur bone. Auto CAD was used to design the patient-specific implant. ANSYS software was used to carry out finite element analysis (FEA) on the designed implant for various materials. By carrying out a structural analysis of the designed implant, we can get to know about the mechanical strength of the designed implant for various materials. This pre-simulation of the hip implant is very useful in determining the stability and compatibility of the implant to be placed into the patient's body. The femur is one of the strongest bones of the human body. Hence, replacing the proximal part of the femur bone with an appropriate implant is very crucial and necessary in order to overcome the problem of revision arthroplasty, which is a very painful, time consuming, and an expensive procedure. In order to overcome these problems, it is better to simulate the loading and unloading stress on the designed femur implant. The stability of the implant can be increased by adding pores in it in order to facilitate the movement of the cells and plasma fluid through it, which may in turn, lead to the growth of new cells, as well as new tissue growth and formation. Hence, we performed FEA on the two femur models of the implant in which one was normal without any pores (NWP), one with pores of varying diameter (one at the proximal end and the other at the distal end). The holes were placed a few centimeters below the neck of the proximal femoral implant. The comparison of the models with and without pores has been shown in the result. The general materials used for the pre-simulation of implants include Stainless Steel, Titanium alloy (Ti6Al4V) and Elgiloy. The overall comparisons of mechanical strength of the materials and designs have been tabulated and represented graphically in the result section.

2. METHODOLOGY

2.1 OBTAINING 3D CT MODEL USING MIMICS:

The total femur bone CT model was developed using MIMICS software as shown in Figure-1 (Materialise). Materialise Mimics is an image processing software for 3D designs and modeling. A CT image of the femur was used to design the patient-specific total femur replacement model which provided a dimension for the implant to be designed. A total of two 3D models of the femur bones were extracted and stored as STL.files. The femur was measured and its dimensions were obtained. Both the 3D models of bones were taken from healthy individuals. An inbuilt software called 3D builder was used to open the 3D model of the femur bone. Mimics were used to measure the dimensions of the femur bone. The measured dimensions were averaged and then the resultant dimensions were used to design the implant.

Fig -1: Proximal Femur bone segmented using Mimics Software

2.2 DESIGNING TOTAL FEMUR IMPLANT USING AUTOCAD:

The hip-implant was designed using AutoCAD software. AutoCAD is a mercantile computer-aided design(CAD) and drafting engineering software. The dimensions of the femur bone were measured using the 3D model obtained using mimics. In this research, a total of two implants were designed as shown in Figure-2. The designed hip replacement implant models are saved as 'iges.file' format. Both the implant models have a femoral stem length of about 8.2cm, but the head (ball) diameters vary in both the designs [11]. Design '1' has a femoral head (ball) diameter of 2.3cm and design '2' has a femoral head diameter of 3cm [12]. Design '2' has pores at the proximal and distal end in order to facilitate osseointegration. The pore at the proximal end has a diameter of 7mm, and the pore at distal end has a of diameter 5 mm. The pores were cut through the implant in a horizontal plane in varying diameters.

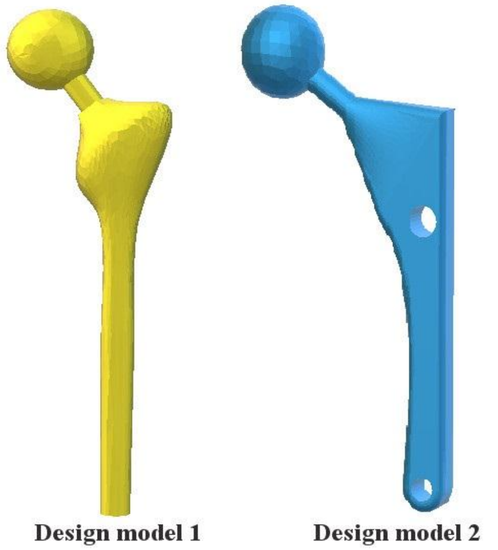

Fig -2: THR models produced with varying design using AutoCAD

2.3 STRUCTURAL ANALYSIS USING ANSYS:

The designed implant has been tested using ANSYS. ANSYS software is used to design 3D models and objects, as well as to perform simulations that test a product's endurance, distribution of temperature, fluid dynamics, and electromagnetic capability. The designed implant was tested for its static structural analysis in ANSYS. It carries out the FEA method, which is a numeric method for solving the engineering and mathematical physics problems and equations. The implant models were imported into ANSYS 18.1 in iges.file format. The implant models were tested for their mechanical strength. The models were examined for the material Ti6Al4V, Stainless Steel, and Elgiloy. Ti6Al4V is widely used for the fabrication of various implants, and is the most stable material; it also promotes osseointegration [13]. The femoral stem was given as fixed support. Forces were applied on the implants head from '3' different axis. The joint contact forces applied were 'x' axis – 433.8N, 'y' axis – 263.8N, 'z' axis – 1841.3N [14].

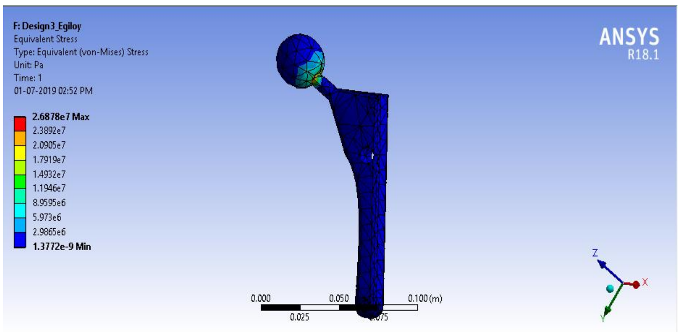

Fig -3: Von Mises Stress result of THR Design-2 of Egiloy material obtained by Ansys

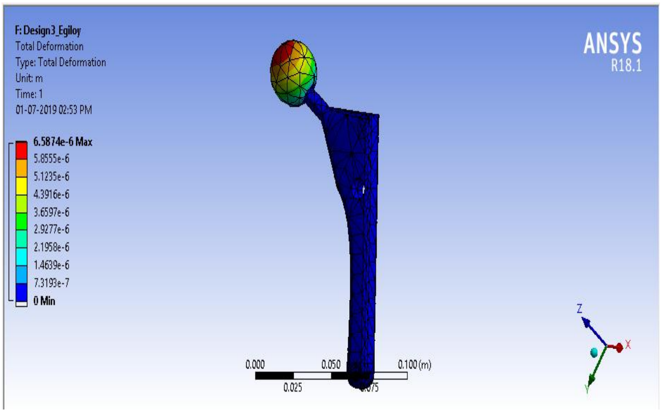

Fig -4: Total Deformations Finite Element Analysis result of THR Design-2 of Egiloy material obtained by Ansys

3. RESULTS AND DISCUSSION

3.1. Finite element analysis

Finite element analysis is used to test compressive and effective tensile modulus of different stent geometries and shapes. The Finite element analysis data generally has a 30% error. The result was obtained and analysed in terms of von-mises stress and total deformation. The proximal part of the implant (neck of the implant) experienced higher magnitude of stress. The medial and distal part of the implant experienced the minimum amount of stress as shown in Figure-3. The von-mises stress and total deformation are tabulated and a graph has been plotted for the same.

Table -1: Total Deformations in (mm) values obtained by FEA analysis

	Stainless Steel 316L	Ti6Al4V	Elgiloy
Design 1	1.00E-04	1.74E-04	1.04E-04
Design 2	6.34E-06	1.12E-05	6.59E-06

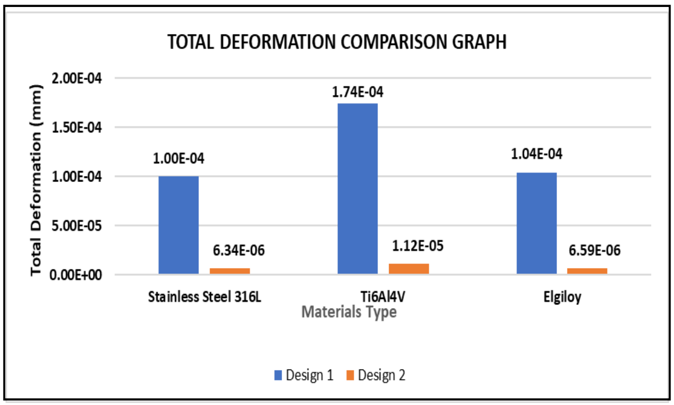

Chart -1: Total Deformations comparison result obtained by FEA analysis

Table -1: Equivalent Von-mises Stress (MPa) values obtained by FEA analysis

	Stainless Steel 316L	Ti6Al4V	Elgiloy
Design 1	289.44	279.33	284.73
Design 2	23.98	23.5	26.87

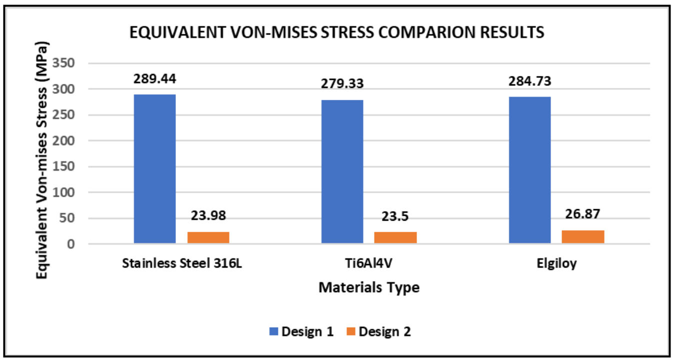

Chart -2: Equivalent Von-mises Stress (MPa comparison results obtained by FEA analysis

4. CONCLUSION

In this study two hip prosthesis were designed and FEA was performed to determine their mechanical stability. In both the implants maximum deformation was observed at the ball region. Maximum deformation was noticed at the ball region because joint contact force was applied on to the ball directly. Slightly high stress was observed at the neck region. This has been attributed to the thin design of the neck. The stems of both implants experienced the least amount of stress and underwent minimum of deformation. The magnitude of stress varied from the head region to the femoral stem region. The whole stress distribution was even throughout the implant which is essential for osseointegration (bone growth) and for the implant to stay intact in the femoral bone. The magnitude of stress varied drastically between both the implants. Design 1 shows higher magnitude of stress when compared to design 2, but when it comes to total displacement design 2 showed the least amount of deformation when compared to design 1. This is due to a thicker stem in design 2. Not much difference was observed in the magnitude of stress when compared with different materials used in the study. Elgiloy is a new material which is extensively used for the manufacturing of implants due to its properties such as corrosion resistance and high strength [15].

FEA is a dependable method which can be implemented in implant analysis and testing to determine the mechanical stability of any implant. The major objective was to lower the stress and displacement which has been achieved in design '2' by making pores and providing a curved neck.

5. Reference

[1] Ridzwan, M. I. Z., Shuib, S., Hassan, A. Y., Shokri, A. A., & Ibrahim, M. M. (2007). Problem of stress shielding and improvement to the hip implant designs: a review. *J. Med. Sci*, *7*(3), 460-467.

[2] Derar, H., & Shahinpoor, M. (2015). Recent patents and designs on hip replacement prostheses. *The open biomedical engineering journal*, *9*, 92.

[3] Shuib, S., Sahari, B., Shokri, A. A., & Chai, C. S. (2008). The design improvement of hip implant for Total Hip Replacement (THR). *Jurnal Kejuruteraan*, *20*, 107-113.

[4] Colic, K., & Sedmak, A. (2016). The current approach to research and design of the artificial hip prosthesis: a review. *Rheumatol Orthop Med*, *1*, 1-7.

[5] Abraham, R., & Malkani, A. L. (2005, June). Instability after total hip replacement. In *Seminars in Arthroplasty* (Vol. 16, No. 2, pp. 132-141). WB Saunders.

[6] Viceconti, M., Brusi, G., Pancanti, A., & Cristofolini, L. (2006). Primary stability of an anatomical cementless hip stem: a statistical analysis. *Journal of biomechanics*, *39*(7), 1169-1179.

[7] Zagra, L. (2017). Advances in hip arthroplasty surgery: what is justified?. *EFORT open reviews*, *2*(5), 171-178.

[8] Ruff, C., Holt, B., & Trinkaus, E. (2006). Who's afraid of the big bad Wolff?:"Wolff's law" and bone functional adaptation. *American Journal of Physical Anthropology: The Official Publication of the American Association of Physical Anthropologists*, *129*(4), 484-498.

[9] Sumner, D. R. (2015). Long-term implant fixation and stress-shielding in total hip replacement. *Journal of biomechanics*, *48*(5), 797-800.

[10] Park, J. B. (2000). Hip joint prosthesis fixation: problems and possible solutions. *Biomedical Engineering Handbook*.

[11] Kim, J. T., & Yoo, J. J. (2016). Implant design in cementless hip arthroplasty. *Hip & pelvis*, *28*(2), 65-75.

[12] Tsikandylakis, G., Mohaddes, M., Cnudde, P., Eskelinen, A., Kärrholm, J., & Rolfson, O. (2018). Head size in primary total hip arthroplasty. *EFORT open reviews*, *3*(5), 225-231.

[13] Colic, K., Sedmak, A., Grbovic, A., Tatic, U., Sedmak, S., & Djordjevic, B. (2016). Finite element modeling of hip implant static loading. *Procedia Engineering*, *149*, 257-262.

[14] Abdullah, A. H., MohdAsri, M. N., Alias, M. S., & Tardan, G. (2010). Finite element analysis of cemented Hip arthroplasty: influence of stem tapers. In *Proceedings of the international Multi Conference of engineering and computer scientists*.

[15] https://www.elgiloy.com/wire-division/wire-elgiloy-alloy/