Simulation on Hip Implant Using Finite Element Method

Amar Reezlan Amar Jauhari[1], Haslina Abdullah[1]*

[1] Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia,
86400, Johor, MALAYSIA

*Corresponding Author: haslinaa@uthm.edu.my

Abstract

Hip implant is used to replace damaged bone on hip joint. The stem design is important since it will influence the performance of hip arthroplasty. In this study, simulations has been performed to investigate the effect of different geometry of implant to the stress distribution and displacement magnitude using the finite element method. The femoral bone was constructed using scan topography while the implant was from CAD software. In this study, there are three types of implant designs have been constructed which is stem without feature, stem with flute and stem with fin design. Then, Abaqus software has been employed to perform the analysis of stability and stress distribution. A hip implant was inserted into the femoral bone in Finite Element Method software and a simulation of displacement and stress distribution was conducted under walking conditions. Based on the simulation, it has been found that the fin feature developed the lowest displacement magnitude and stress distribution which are 0.07 μm and 58.81 Pa. As conclusion, adding additional feature to the implant would improve the implant's stability while also experiencing less stress distribution

1. Introduction

Total hip arthroplasty, also be recognized as hip replacement surgery, is widely regarded as one of the most significant advances in orthopedics in the twentieth century. It is used to treat hip joints that have been stricken by osteoarthritis [1]. Hip joints must be replaced by an artificial implant, and total hip arthroplasty has been shown to reduce pain and improve function in both acute and chronic situations [2]. Afterwards, patients can resume typical daily activities with normal range of motion [3].

Several studies have tried to improve current hip arthroplasty implants function. This includes investigating several adjustments for current use of implant in hip arthroplasty using Finite Element Analysis, where with more resources, more modifications can be done in real life to improve the design, and the application of Finite Element Method (FEM) is reliable to provide preliminary results for this approach [4]. A study investigates the impact of cutting flute design on implant body to determine the

2. Methodology

The femoral bone used was of the general bone of Asian people. The material properties for both hip implant and femoral bone are shown in Table 1 [6]. In this study, a model of hip implant has been developed using Solidwork software. The model of femur bone was created using scan topology of Asian patient. The models will be exported to Abaqus software to examine the stresses acting on the implant and also its magnitude of displacement. In Abaqus software also,the material of hip implant and femoral bone has been defined. Several parameters need to be assigned in before conducting the simulation such as the properties for the parts,

boundary condition and the loads acting on the implant and femur bone need to be determine. The simulation data will be validated and compare with previous study.The main purpose of this study is to investigate stress distribution and displacement acting on the implants. Solidworks has been used to design the three design of implants. The general dimension of the implant used is 15mm to 8mm wide and 160mm long and it is made up from material Inconel 718 [7]. Material parameters for the implant are shown in Figure

Table 1 *Material properties of bone and implant*

Item	Elastic Modulus (GPa)	Poisson's Ratio
Implant	200	0.29
Femur bone	17.3	0.3

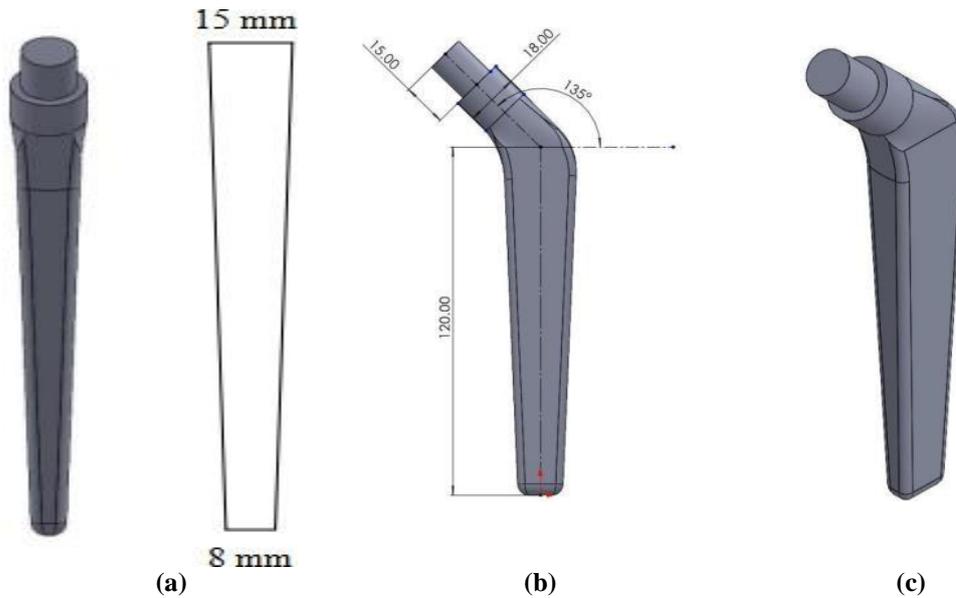**(a)**   **(b)**   **(c)**

Fig. 1 *General parameters for implant design (a) Front view (b) Side view (c) Isometric view*

For fluted implant, an inward curve of 3mm deep were applied onto both side of the implant. The side view design of fluted feature is as Figure 2 . To design the fin implant, two layers of fins will be built on both side surfaces below the implant's proximal part. The fins are 2mm wide and 20mm long, with a 1mm thickness. The side view of fin feature is as Figure 3.
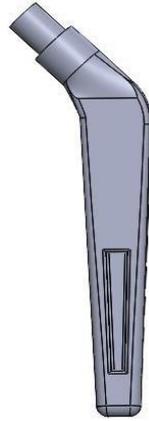
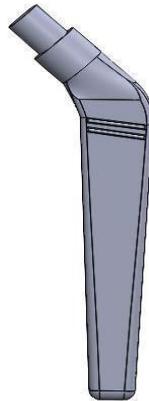**Fig. 2** *Side view of flute implant*

Fig. 3 *Front view of flute implant*

After completing the design of the three features, the implants was imported and merged into femoral bone using Abaqus software. Before start the simulation,the material properties of the implants were assigned as shown in Table 1. For this study, it is a static study of where no movement created in any axis. Step 1 will be applied in the process. Furthermore, field output and history output were determined. The field output request for this study focusing on stresses and the displacements occurred after applying loads. The boundary condition that are assigned to the implant hold the models without any rotation acting on any axis implying the models are static.The boundary condition set for the cutting tool are Encastre (U1=U2=U3=UR1=UR2=UR3) with all the value are set as 0 respectively. The loads were assigned onto the model at specific points to copy normal walking condition as shown in Figure 4 [8]. Table 2 shows the value of forces acted during walking condition in xyz-directions.
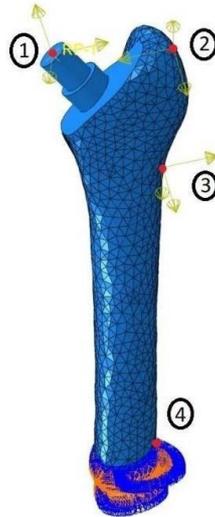
Fig. 4 *Point of forces contacting the model*

Table 2 *Value of forces contacting the model*

Point	Forces in x-direction (N)	Forces in y-direction (N)	Forces in z-direction (N)
1	433.8	263.8	-1841.3
2	-403.8	-128.3	9.6
3	7.7	-159	-798.9
4	0	0	0

Next is the meshing part of the implant and femur bone. The meshing type used for the complex design was linear tetrahedral. Figure 5 shows the completed mesh of the model. Under job module, a job file was created to obtain the data required.
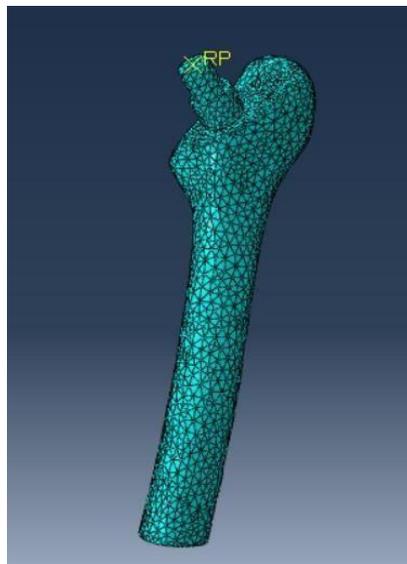

Fig. 5 *Point of forces contacting the model*

3. Results and Discussion

Analysis of the simulation results includes the stress distribution and displacement magnitude occurred on the implant for the three designs. Results are obtained from the finite element method using Abaqus software. The purpose of this study is to identify the effect of additional features to the stress distribution and displacement magnitude.

3.1 Relative Displacement

For displacement magnitude, the threshold value of 40-150 µm for relative displacement was used [9]. The value of below 40 µm promotes the osteointegration to occur and enhances the rate of bone growth to the implant surface [10]. Furthermore, value above 150 µm will led to the formation of higher fibrous tissues on the bone thus causing loosening of implant [11]. Figure 6 shows the comparison of relative displacement for the three implants.
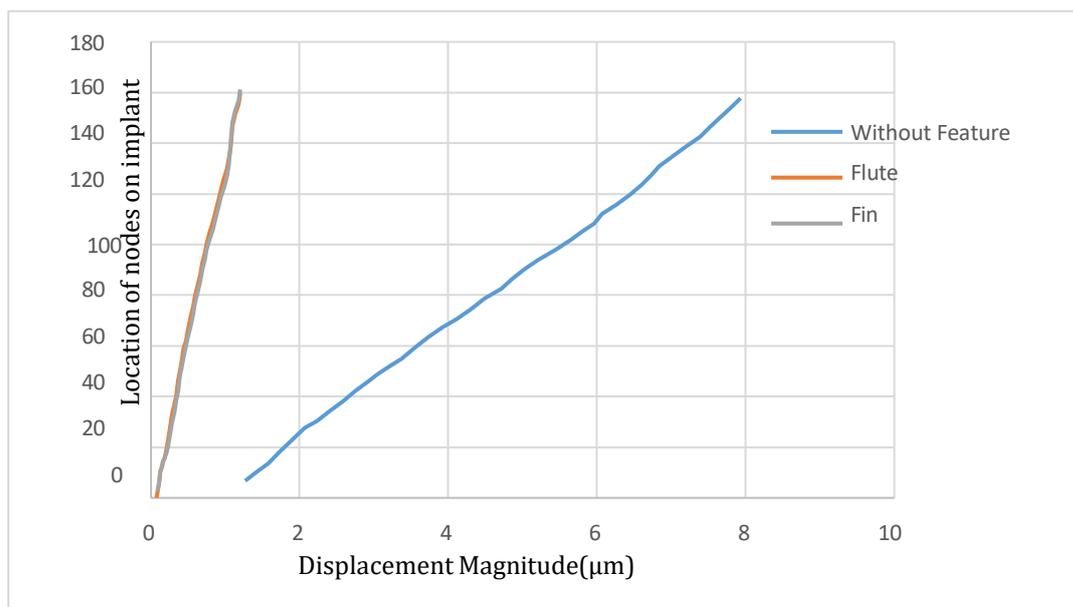

Fig. 5 *Displacement magnitude of original, flute and fin design*

It can be observed that fin implant produced the lowest relative displacement among the other two designs which are 1.20 µm for fin, 1.25 µm for flute, and 7.93 µm for original design. It concluded the design were also below the threshold value of 40 µm thus osteointegration will occur, enhancing the rate of bone growth to the implant surface. The result implies that creating fin design onto original implant would be beneficial as it conveys less displacement magnitude and also contributes to the better rate of bone growth on implant surface.

3.2 Stress Distribution

The results of this study focus on the stress distribution on the bone-implant interface. Relative stresses refers to the amount of loading occurred onto the implant on the selected nodes. The results' stress distribution will be compared with the implant's material Young's Modulus of 200 Gpa to determine whether the additional geometry will fail or not after applying loads. Figure 6 shows the comparison of stress distribution among the three designs.
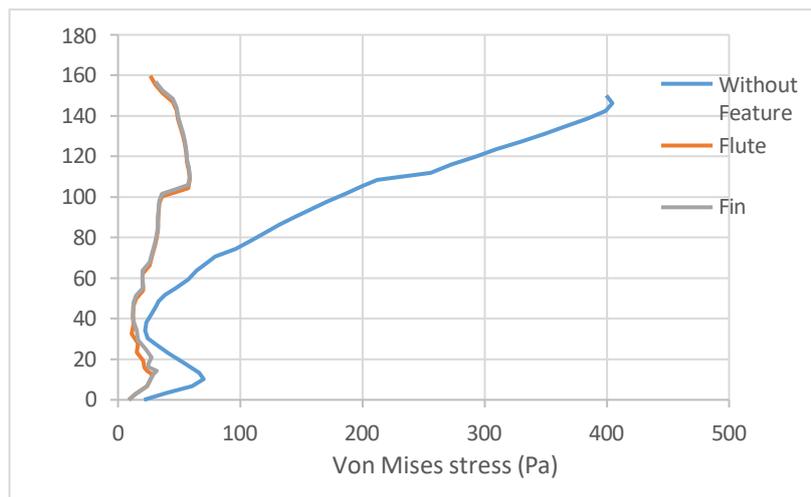
Fig. 6 *Stress magnitude of original, flute and fin design*

4. Conclusion

Initial stability of the hip implant for total hip arthroplasty was obtained by its relative displacement of node on implant surface to the node on the femur bone by using finite element method, under normal walking condition. The threshold value used in this study was 40 μm - 150 μm. Simulations were carried out to investigate on the effect of having fluted and fin design to the displacement magnitude. As a result, it was found that implementing both designs would improve the implant's stability, with flute and fin have 1.25 μm and 1.20 μm respectively. These magnitudes are lower than the threshold limit of between 40 to 150 m. The highest value of relative displacement was lower than the minimum threshold value and was believed that it can lead to osteointegration to occur, enhances the rate of bone growth to the implant surface. Simulations were also carried out to study the effect of having fluted and fin design on value of stresses occurred on implant and compared the results along the initial design with the Young's Modulus of the implant which is 200 GPa. Simulations show maximum stresses occurred on initial design was 455.2 Pa, while both flute and fin design display much lesser value, which are 71.22 Pa and 58.88 Pa, respectively. Implant with fin feature is the best choice among the other options as it has the lowest initial stability displacement and stress distribution.

Acknowledgement

References

[1] Chu, C. (2015). Short-term Analysis vs Long-term Data on Total Hip Replacement Survivorship. JAMA Surgery, 150(10), 989.

[2] Li, M., Venäläinen, M., Chandra, S., Patel, R., Fripp, J., & Engstrom, C. et al. (2021). Discrete element and finite element methods provide similar estimations for hip joint contact mechanics during walking gait. Journal Of Biomechanics, 115, 110163.

[3] Chethan, K., Shyamasunder Bhat, N., Zuber, M., & Satish Shenoy, B. (2021). Finite element analysis of hip implant with varying in taper neck lengths under static loading conditions. Computer Methods And Programs In Biomedicine, 208, 106273.

[4] 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

[5] Jimbo, R., Tovar, N., Marin, C., Teixeira, H. S., Anchieta, R. B., Silveira,L. M., Janal, M. N., Shibli, J. A., & Coelho, P. G. (2014). The impact of a modified cutting flute implant design on osseointegration. International Journal of Oral and Maxillofacial Surgery, 43(7), 883– 88

[6] Fox, G. R., & Liang, H. (2010). Wear mode comparison of high-performance Inconel Alloys. Journal of Tribology, 132(2).

[7] Kladovasilakis, N., Tsongas, K., & Tzetzis, D. (2020). Finite Element Analysis of Orthopedic Hip Implant with Functionally Graded Bioinspired Lattice Structures. Biomimetics, 5(3), 44.

[8] Abdullah, H. (2012). The effect of geometry of prosthesis on the stability of hip joint arthroplasty using finite element method.

[9] Staedt, H., Kämmerer, P. W., Goetze, E., Thiem, D. G., Al-Nawas, B., & Heimes, D. (2020). Implant primary stability depending on protocol and insertion mode — an ex vivo study. International Journal of Implant Dentistry, 6(1). https://doi.org/10.1186/s40729-020-00245-3

[10] Kohli, N., Stoddart, J. C., & van Arkel, R. J. (2021). The limit of tolerable micromotion for implant osseointegration: A systematic review. Scientific Reports, 11(1).