Computational Design of Prosthetic Hip Implant

Abdulrahman AL-sanea, Akram Omara, Nitturi Naresh Kumar, Megdi Eltayeb
Biomedical Engineering Department Sudan University of Sciences and Technology
alsane10@gmail.com[1]

Recieved: 12/12/2018
Accepted: 15/01/2019

Abstract- In this study a 2D/3D CAD model of an artificial hip joint was designed using SolidWorks software program. Biocompatible and robust enough materials such as Titanium alloy Ti6Al4V and Alumina ceramic Al2O3 were used in the designing process for such parts of the hip joint. The finite element (FE) method was used to obtain a solution for the stress and strain distribution throughout series of adjacent elements, further the design was analyzed using computer-aided engineering (CAE) software after it was loaded with static loads 250N, 350N, and 450N respectively to check the durability of the design. The stability of the model was verified by static test.

Keywords: ***Computational Modeling, Hip joint, Biomaterials, Finite Element, Static Load.***

المستخلص في هذه الدراسة تم تصميم نموذج 2D / 3D CAD لمفصل الورك الاصطناعي باستخدام برنامج SolidWorks. تم استخدام مواد متوافقة حيويا وقوية بما فيه الكفاية مثل سبائك التيتانيوم Ti6Al4V وألومينا السيراميك Al2O3 في عملية تصميم هذه الأجزاء من مفصل الورك. تم استخدام طريقة العناصر المحدودة (FE) للحصول على حل لتوزيع الضغط والإجهاد عبر سلسلة من العناصر المتجاورة ، بالإضافة إلى تحليل التصميم باستخدام برنامج الهندسة بمساعدة الكمبيوتر (CAE) بعد تحميله بأحمال ثابتة 250N ، 350N ، و 450N على التوالي للتحقق من متانة التصميم. تم التحقق من استقرار النموذج عن طريق اختبار ثابت (استاتيكي).

INTRODUCTION

Hip joint is one of the major joints in human body which deteriorate with time for many problems. Some people with hip problems need to artificial hip joints to help them in mobility. The number of individuals undergoing hip replacement surgery has accumulated over the past decades. Within the United Kingdom alone, quite sixty thousand total hip arthroplasties (THA) area unit performed annually. Out of all these 9000 area unit performed within the younger people (less than fifty seven years old [1]). Most hip replacements area unit performed on patients plagued by arthritis, atrophic arthritis and vascular necrosis [2].

The artificial hip joint is made of two main parts. The acetabular component (socket) replaces the acetabulum in the pelvis and the femoral component such as stem and ball head put in place of the femoral head [3]. There are four combinations of femoral head and acetabular components. These are taken into consideration depending on the specifications as each combination has pros and cons based on patient condition doctor select the suitable implant. The first and second classification is metal-on-metal and metal-on-polyethylene respectively. The third and fourth classification is ceramic-on-ceramic and ceramic-on-polyethylene respectively. The metal-on-polyethylene is mostly preferred choice. By adding nanotubes / graphen in to polyethylene is revealed improved mechanical properties of polyethylene and also increased in longevity of the implant[4, 5]. Surface coating, gamma irradiation and plasma treatment of polyethylene is alternative methods to improve surface properties [6, 7].

One of the factor that effect on hip joint performance is friction; it differs according to the type of combination. In our study we selected ceramic-on-ceramic combination because it has lowest friction comparing with the other types of combination. As well as the results of up-to-date alumina-on-alumina THA with a metal-backed socket and a cementless stem were excellent after 5-year minimum follow-up. What is more, we have a tendency to believe modern alumina-on-alumina bearing supply a promising possibility for younger, active patients with osteonecrosis of the femoral head [8].

Considerable dimensions and geometric accuracy were required in the design and any inaccuracy in the design will not be accepted to manufacture, so the dimensions of the design were selected according to the ISO 7206-part 1 for classification and designation of dimensions [9].

The design and the final inspection of these implants have play vital role in whole procedure of the manufacturing, and of course in their tendency to failure. For this reason authoritarian international standards were instituted to protect and make reliable the design and the machining of them [10].

A three-dimensional model of total hip was framed such as to help in simulation of location of both the femoral and acetabular components. The model is summarized as shown in Figure1.

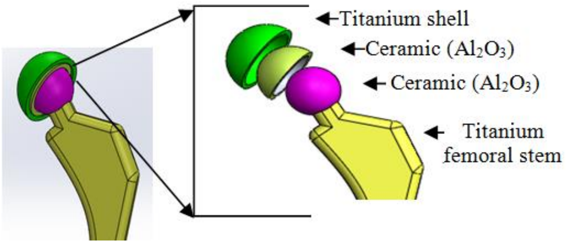


Figure 1: A three-dimensional model of total artificial hip components.

Methodology

SolidWorks software program (version 2016) was used for the modeling and simulation of an artificial hip joint. The finite element (FE) method was used to obtain a solution for the stress and strain distribution throughout series of adjacent elements when the applied loads, boundary conditions and material properties are known, later the design was analyzed using computer-aided engineering (CAE) software. The following sections describe the design process and the loading conditions of the total hip components. The modeling and simulation processes of an artificial hip joint are shown in Figure4.

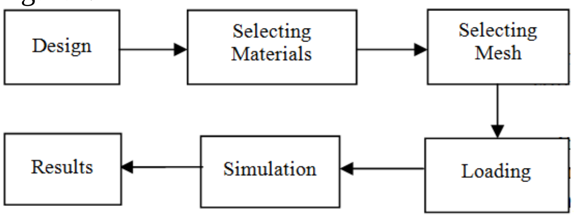


Figure 4: Modeling and simulation processes of total artificial hip joint components.

A. *Acetabular Component Design*

The acetabular component (socket) was designed using SolidWorks software program. It was made of metal shell with five screws and inner liner of Alumina ceramic Al_2O_3 that provides the bearing surface with diameters of the inner surface of the metal shell was 38 mm and outer surface of the Alumina ceramic Al_2O_3 was and 37 mm. This structure allows 0.5 mm between them. The outer diameter of the acetabular component (socket) was assumed to be 44 mm. As well as the diameters of the ball head and the inner surface of the Alumina ceramic Al_2O_3 were 30 mm and 31 mm respectively, giving a distance of 0.5 mm between the ball head and the inner surface of the Alumina ceramic Al_2O_3. The design of an acetabular component is shown in Figure2, and its dimensions are shown in Figure 3.

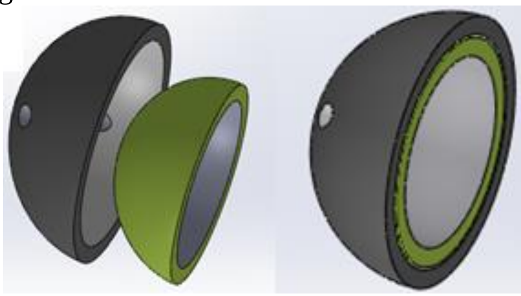


Figure 2: CAD model of an Acetabular Component using SolidWorks software.

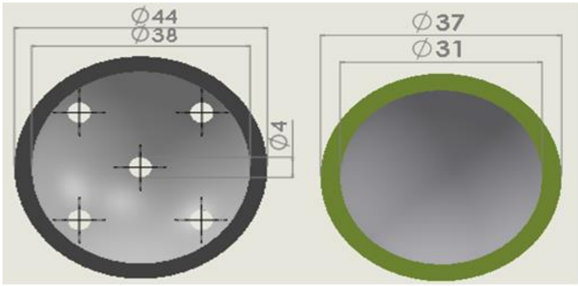


Figure 3: Acetabular component dimensions, metal shell dimensions on the left, and inner liner dimensions on the right.

B. *Femoral Component Design*

The 3D CAD model of femoral component was design using SolidWorks software program as shown in Figure 5.

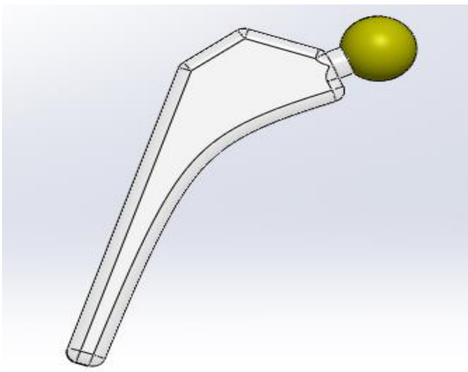


Figure 5: CAD model of femoral component design using SolidWorks software.

It was made of stem and ball head that connect the stem with the acetabular component. The femoral stem length was 160 mm, the width was 46.21 mm, the neck length was 11.77 mm, the neck diameter was 15.48 mm, the thickness of the stem was 14 mm, and ball head diameter was

30 mm. All dimensions of the femoral component design are shown in Figure 6.

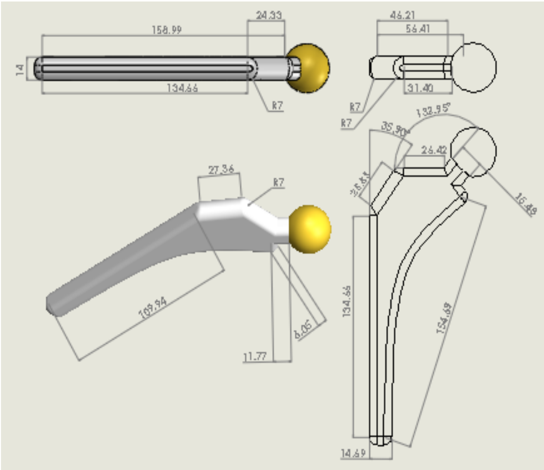


Figure 6: Femoral component dimensions.

C. Loading Conditions

After the design of all parts of hip joint, CAE was used to do a number of static analysis and examination for all parts after loading them with static loads 250N, 350N, and 450N in order to see if these loads through analysis are in the frames of standards. The analysis was performed on the most important parts of the hip joint, the femoral head according to the ISO 7206-part 10 and acetabular component according to the ISO 7206-part 12 [11].

Static analysis was carried out by using SolidWorks software program (version 2016) for both models of femoral and acetabular components, these models simulate a human during upright standing. When weight is equally distributed across each legs throughout upright standing, the burden supported at every hip is simple fraction the burden of the body segments higher than the hip, or concerning third of total body weight [12]. Load applied on hip joint is 250 N for 75 kg weight person. The femoral stem and metal shell were assumed to be fixed, and hence whole load falls the center of the femoral head and inner surface of Al_2O_3.

There are three parameters effect on hip joint performance; these parameters are the load (F), area of femoral head (A) and length of femoral stem (L). As the femoral stem of hip joint is fixed and the area of hip joint is known the stress (б) of hip joint is directly proportional with load as shown in equation (1) below:

$$б = F/A \quad (1)$$

The area (A) of femoral head is calculated from the equation of ball area (r = 15mm): A = $4\pi r^2$. Also, the strain (Ɛ) is directly proportional with load; as the length of femoral stem (L) is constant, when the load is increased the change in length (ΔL) is increased, thus the strain increased as shown in equation (2) below:

$$Ɛ = \Delta L/L \quad (2)$$

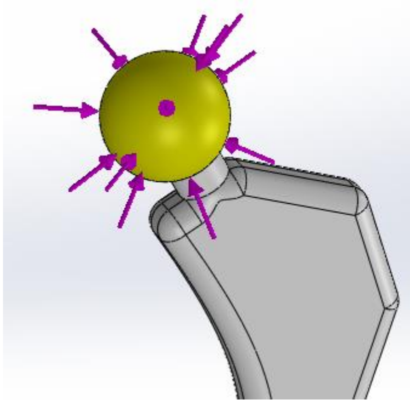


Figure7: The distribution of force on the femoral head

Materials and Method

The materials were selected according to the standards, the acetabular component was made of a metal shell of Titanium alloy Ti6Al4V with an inner liner was made of Alumina ceramic Al_2O_3 that provides the bearing surface and the femoral stem was made of Titanium alloy Ti6Al4V as in shell in acetabular component and the ball head was made of Alumina ceramic Al_2O_3. The important material parameters required for the FEM analysis are the elastic modulus and the Poisson ratio [13]. Table one illustrate properties for the materials in this study. Here modeling has been done as linear elastic isotropic.

The total number of elements for the FE model of femoral component was approximately 62704 elements with element size 2 mm and the total number of nodes was 92415. For the FE model of ace tabular component the total number of elements was 13068 elements with element size 2 mm and the total number of nodes was 22675.

TABLE 1: MATERIALS PROPERTIES

Components	Materials	elastic modulus (GPa)	Poisson's ratio
Femoral stem and Metal shell	Titanium alloy Ti6Al4V	110	0.31
Femoral head Inner liner	Alumina ceramic Al2O3	380	0.22

We used the solid tripartite mesh because it's faster than the quadratic mesh. The used mesh types which are shown in Figure 8.

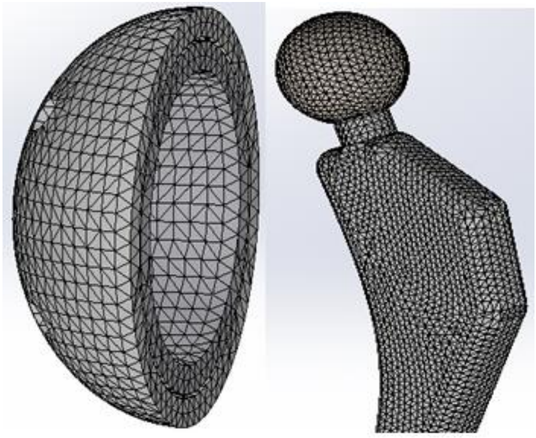


Figure 8: Femoral and ace-tabular components with solid tripartite mesh.

Results and Discussion

Figure 9, (a, b, c), Figure 10, (e, f, g), and Figure 11, (i, j, k) show the stresses, strain, and displacement of femoral head under static load 250N, 350N and 450N, respectively. The red and green colors at the connected point between the ball head and the top edge of the neck represent the critical area because most of the applied force operates in this area.

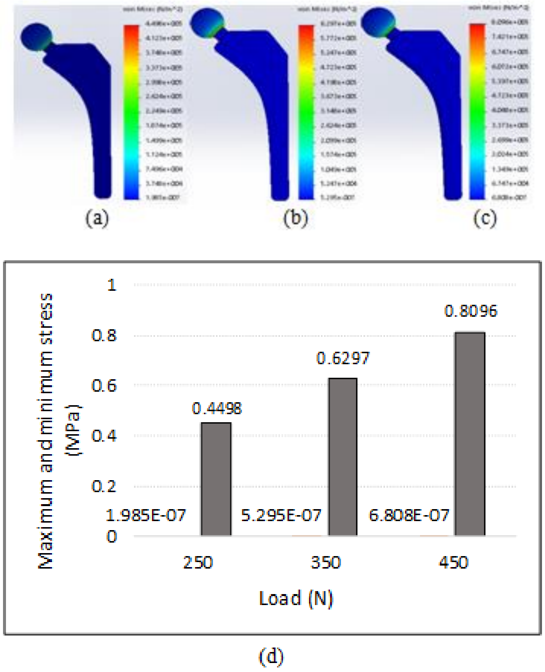


Figure9: (a), (b) and (c) Maximum and minimum stresses under static loads 250N, 350N and 450N, respectively, (d) shows the histogram of stress analysis.

Figure 12, (m, n, o), Figure 13, (p, q, r) and Figure 14, (s, t, u), show the stresses, strain, and displacement of acetabular component under static load 250N, 350N and 450N, respectively. The maximum stress and strain distribute in the inner surface of Alumina ceramic because of the reaction force of the ball head during standing distributes in all the inner surface of Alumina ceramic Al_2O_3, and the maximum displacement covers the all area in the inner surface of Alumina ceramic Al_2O_3 because a distance of 0.5 mm between the outer diameter of the Alumina ceramic Al_2O_3 and the metal shell.

The results of maximum and minimum stresses, strain, and displacement of femoral head under static loads 250N, 350N and 450N are shown in Figure 9, (d), Figure10, (h) and Figure11, (l), comparing the three static loads the stress, strain and displacement of femoral head are directly proportional. As stress proportional with strain, in the area where the stresses are on critical value, there is a large strain, and from that area the crack of the joint will be occurred, and this is will lead to total damage of the hip joint.

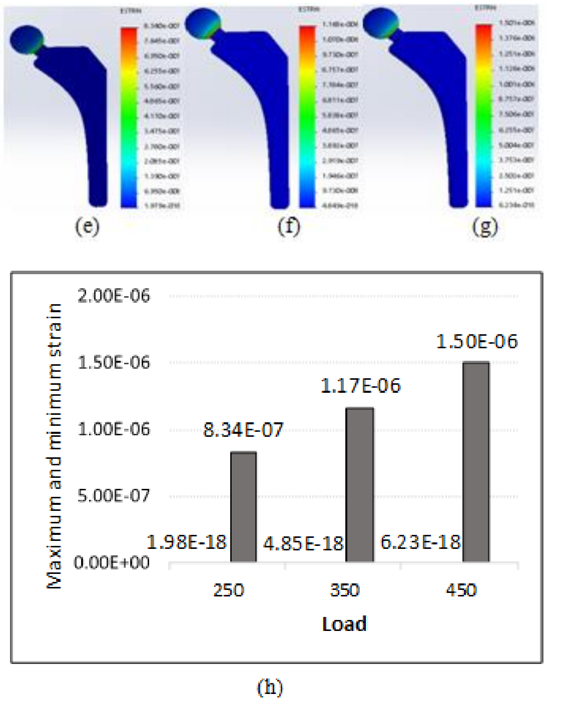
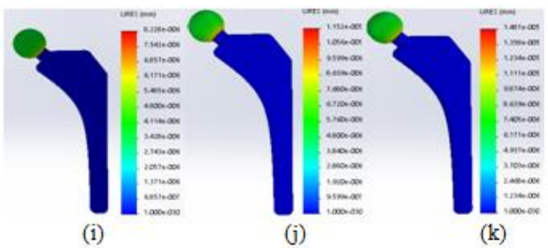
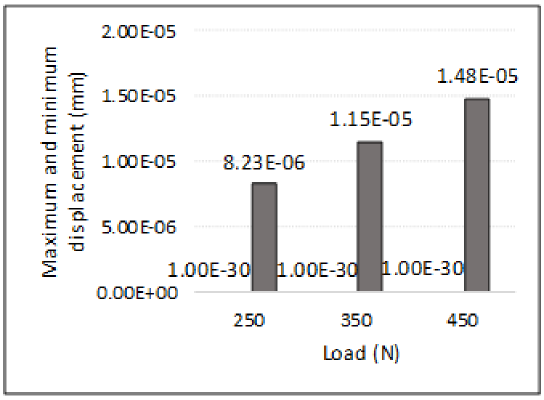


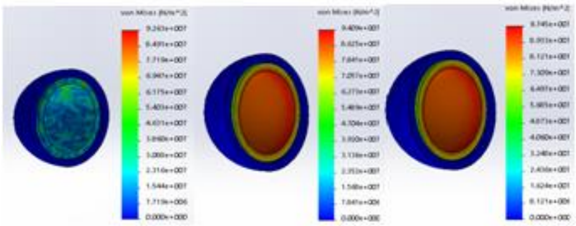
Figure 10: (e), (f) and (g) Maximum and minimum strain under static loads 250N, 350N and 450N, respectively, (h) shows the histogram of strain analysis.





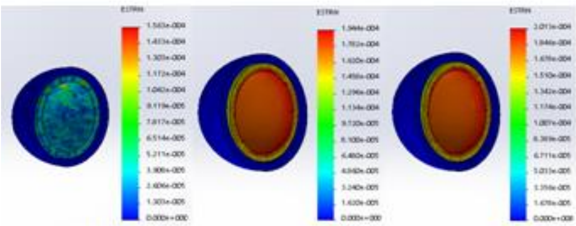
(l)

Figure11. (i), (j) and (k) Maximum and minimum displacement under static loads 250N, 350N and 450N, respectively, (l) shows the histogram of displacement analysis.



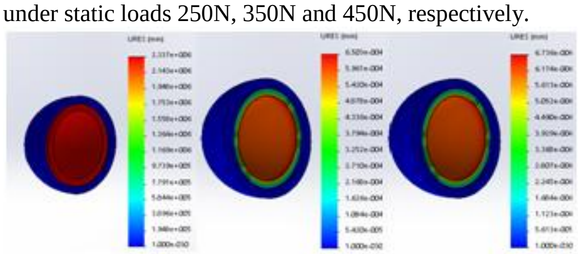
(m) (n) (o)

Figure12: (m), (n) and (o) Maximum and minimum stresses under static loads 250N, 350N and 450N, respectively.



(p) (q) (r)

Figure13: (p), (q) and (r) Maximum and minimum strain under static loads 250N, 350N and 450N, respectively.



(s) (t) (u)

Figure14: (s), (t) and (u) Maximum and minimum displacement under static loads 250N, 350N and 450N, respectively.

Conclusion

This paper summarizes the use of the software modeling in the design and analysis of an artificial hip joint. SolidWorks software was used in this design and analysis procedures of all parts of hip implant. Titanium alloy Ti6Al4V and Alumina ceramic Al_2O_3 were used in the designing process for all parts of the hip joint, then hip joint is static examined using FE method.

Von Mises stress, strain, and displacement distribution are acquired from above examination results are within the acceptable range (blue and green colors), but if the static load was increased to values higher than 450N the stress, strain and displacement of femoral head are directly proportional with the static load so the risk ratio of hip joint damage will increase. Thus, it is terminated that hip joint style investigated during this study area unit safe against fracture failure, and may be factory-made in Sudan.

References

[1] T. Tennent and N. Goddard, "Current attitudes to total hip replacement in the younger patient: results of a national survey," *Annals of The Royal College of Surgeons of England*, vol. 82, p. 33, 2000.

[2] D. A. Macdonald, "(i) Risks versus rewards of total hip replacement," *Current Orthopaedics*, vol. 12, pp. 229-231, 1998.

[3] "Hip Arthroplasty: Hip Joint Replacement | Houston Methodist http://www.houstonmethodist.org/orthopedics/where-does-it-hurt/hip/artificial-joint-replacement/."

[4] P. Rama Sreekanth, N. Naresh Kumar, S. Arun, and S. Kanagaraj, "Effect of multi-walled carbon nanotubes reinforcement and gamma irradiation on viscoelastic properties of ultra-high molecular weight polyethylene," *Materials Research Innovations*, vol. 20, pp. 198-205, 2016.

[5] P. R. Sreekanth, A. Priyedarshi, and N. N. Kumar, "Comparison of Elastic Modulus of MWCNT Polyethylene Composites using Nanoindentation and Macromechanical Analysis," *Int. J. Curr. Eng. Technol*, vol. 2, pp. 320-324, 2014.

[6] P. Sreekanth, N. N. Kumar, and S. Kanagaraj, "Effect of MWCNT on mechanical properties of γ-irradiated UHMWPE during shelf ageing process," in *Advanced Materials Research*, 2012, pp. 160-163.

[7] N. Naresh Kumar, S. L. Yap, M. Z. Khan, and R. S. Pattela Srinivasa, "Effect of Argon Plasma Treatment on Tribological Properties of UHMWPE/MWCNT Nanocomposites," *Polymers*, vol. 8, p. 295, 2016.

[8] S.-Y. Kim, "Alumina-on-Alumina Total Hip Arthroplasty in Patients with Osteonecrosis less than 50 Years Old," in *Bioceramics and Alternative Bearings in Joint Arthroplasty*, ed: Springer, 2007, pp. 219-219.

[9] "ISO 7206 – part 1: classification and designation of dimensions ".

[10] I. An, M. Papadrakakis, M. Kojic, and V. Papadopoulos, "Design, Numerical Analysis and Manufacturing of Artificial Hip Joint Implants," 2009.

[11] "ISO 7206 – Part 10: Determination of resistance to static load of modular femoral heads, ISO 7206 – Part 12: Deformation test method for acetabular component."
[12] S. J. Hall, *Basic Biomechanics* amazon: McGraw-Hill Higher Education, 2011.
[13] H.-b. Jiang, "Static and dynamic mechanics analysis on artificial hip joints with different interface designs by the finite element method," *Journal of Bionic Engineering*, vol. 4, pp. 123-131, 2007.