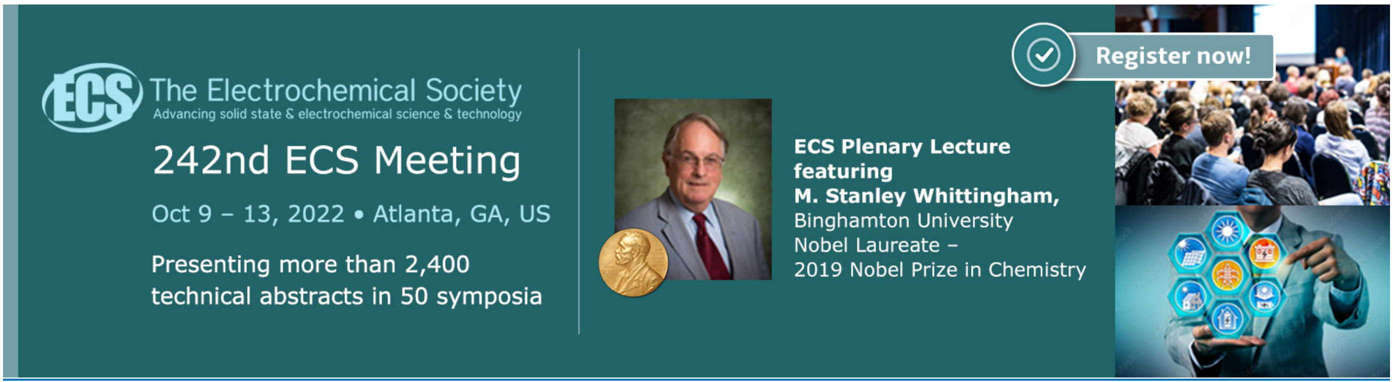
Journal of Physics: Conference Series

Finite element analysis of 2-Station hip himulator

To cite this article: M I M Fazli *et al* 2017 *J. Phys.: Conf. Ser.* **908** 012057



ICADME 2017 IOP Publishing
IOP Conf. Series: Journal of Physics: Conf. Series **908** (2017) 012057 doi:10.1088/1742-6596/908/1/012057

Finite element analysis of 2-Station hip himulator

M I M Fazli[1], A Yahya[2], A Shahrom[2], S W Nawawi[1], M R Zainudin[1] and M S Nazarudin[1]

[1] Faculty of Electrical Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor Bahru, Malaysia
[2] Faculty of Biosciences and Medical Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor Bahru, Malaysia

mikram33@live.utm.my

Abstract. This paper presented the analysis of materials and design architecture of 2-station hip simulator. Hip simulator is a machine used to conduct the joint and wear test of hip prosthetic. In earlier work, the hip simulator was modified and some improvement were made by using SolidWorks software. The simulator consists of 3DOF which controlled by separate stepper motor and a static load that set up by manual method in each station. In this work, finite element analysis (FEA) of hip simulator was implemented to analyse the structure of the design and selected materials used for simulator component. The analysis is completed based on two categories which are safety factor and stress tests. Both design drawing and FEA was done using SolidWorks software. The study of the two categories is performed by applying the peak load up to 4000N on the main frame that is embedded with metal-on-metal hip prosthesis. From FEA, the value of safety factor and degree of stress formation are successfully obtained. All the components exceed the value of 2 for safety factor analysis while the degree of stress formation shows higher value compare to the yield strength of the material. With this results, it provides information regarding part of simulator which are susceptible to destruct. Besides, the results could be used for design improvement and certify the stability of the hip simulator in real application.

1. Introduction

Human body consist of joints which enable every part of the body to move freely up to certain limits. Hip joint or hip articulation is one of important joint in human motion. The hip joint which also known as ball-and-socket joint have large degree of freedom and provide wide range in human mobility. Therefore, dysfunction of this joint will certainly limit the motions of people who suffer from this problem. Hip prosthesis is an orthopaedic procedure which the hip joint is replaced by prosthetic or artificial implant to replace arthritic or dysfunctional joint surface [1]. However, there are lots of postoperative complication occur regarding the hip prosthetic such as dislocation [1], and wear debris [2] which affect the safety and efficacy of patient [3] and eventually reduce the lifespan of hip prosthetic.

Therefore, hip simulator has been developed so that the lifespan of metal-on-metal hips prosthesis can be tested before being implant to the patient by surgery. The hip simulator will generate similar joint posture and joint contact force as in daily life motion. Besides, the simulator will provide useful information regarding the biomechanics of hip joint and evaluated the limits of performance of the biomaterials used in hip implant.

Hip joint simulator is not a recent finding in the field of orthopaedic. Various design of hip simulator has been developed decades ago to regenerate with high accuracy in vivo condition. The simulators differ from each other from a few parameters which are number of station, loading type, sequence of motion, degree of freedom (DOF), type of lubricant, and configuration of hip implant whether it is anatomical or not [4, 5].

The most crucial process in designing mechanical system is to ensure the endurance of the parts and components especially for application that use high force or load. There is very high risk taken by establish a product or system without verify the performance of the structure and materials. Finite element analysis (FEA) is one of the method used by engineer and scientist to model and numerically solve complex structural or system. Development of system or structure in virtual environment will help to predict and enhance the reliability of the product. Besides, it will help in reducing the cost for prototyping and testing which eventually reduce the material usage.

In this paper, structural and materials analysis in term of safety factor and stress formation are implemented on the 2-station hip simulator design using SolidWorks software.

2. Hip simulator model and finite element modelling

Figure 1 shows the design of 2-station hip simulator from different perspective; isometric, front and side. Basically, the simulator is designed with some modification based on specification that are obtained from literature. The system of simulator is expected to replicate human joint motion [6] and provide testing on hip prosthesis in term of wear and dislocation.

Figure 2 represents the principle diagram of main frame in the proposed design hip simulator. The analysis of the simulator in this work will focus only on the main frame which are the upper part and the lower part as shown in figure 3. This is because both parts experience more load than other parts in the simulator. Besides, the hip implant part is embedded in innermost of the frame and become the fixed geometry during the simulation of the whole main frame.

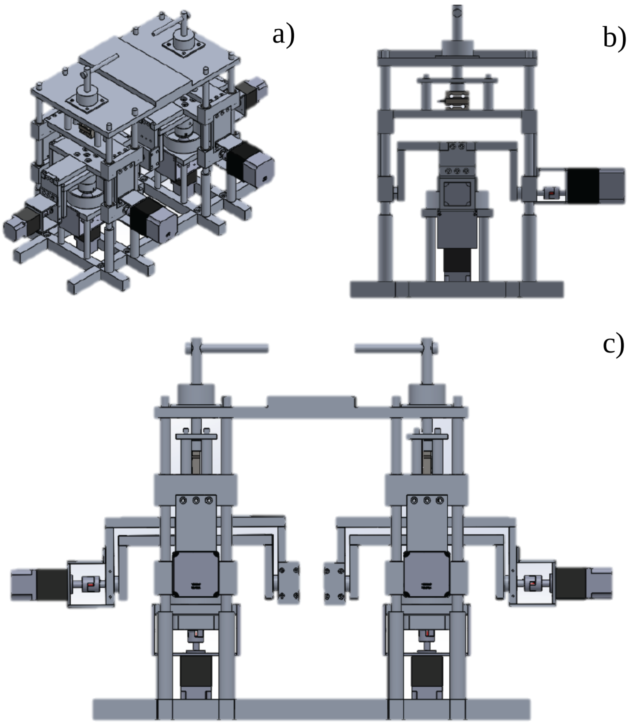


Figure 1. Design of 2-station hip simulator: a) isometric view b) side view c) front view.

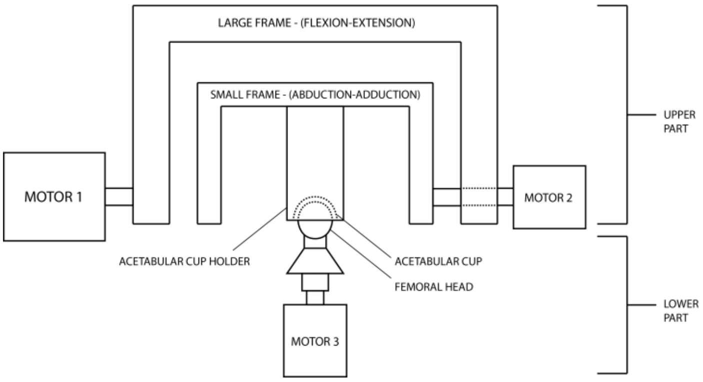


Figure 2. Principle diagram of main frame.

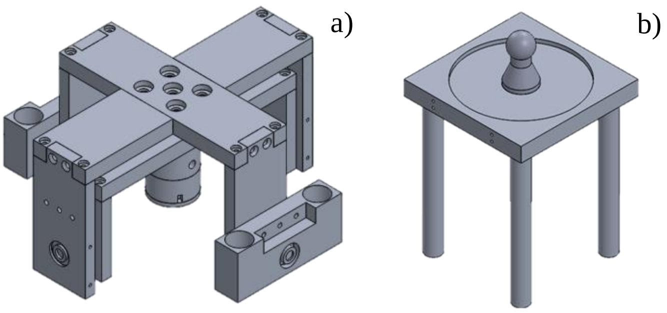


Figure 3. Isometric view a) upper part b) lower part

The SolidWorks software is used to execute the finite element analysis. Aluminium alloys 7050-T73510 is the chosen materials for this analysis and future manufacturing. The details specification of the selected materials is represented in table 1. The stress limit of material can resist without being deformed is indicated by the value of yield strength in table 1.

The analysis start by isolating the main frame model design from the whole model of hip simulator. Next, the materials chosen is applied to all part in the main frame. The fixed geometry is chosen and the load applied in the area shown by purple arrow in figure 4. The value of load is set based on the previous report in [7]. 2000 N load is applied at each side of the holder pushing downward in figure 4 a) while 4000 N load applied to femoral head in figure 4 b). The stress formation and factor of safety (FOS) value is obtained from the analysis. The main frame is tested with variety of state which imitate human hip joint motion such as static state, flexion, extension, abduction, and adduction. The parameter for the motion is taken from graph in [4].

3. Result analysis

The stress formation is defined by looking at the colour formed from the analysis. Red represent the highest deformation followed by yellow, green and the lowest deformation which is blue. Besides, factory of safety result is determined by the number shown in the upper left of the result. If the value of safety factor exceeds 2, the part is found to inherit low deformation scale.

From the simulation, the result for stress formation and FOS were successfully obtained. Based on the stress formation analysis, all the parts in the main frame shows that the frame can withstand high pressure load. Furthermore, the factor of safety value shows value more than 2 for all states.

Table 1. Specification of aluminium alloy 7050-T73510.

	Value	Units
Elastic Modulus	7.2e+010	N/m^2
Poisson's Ratio	0.33	N/A
Shear Modulus	2.69e+010	N/m^2
Density	2830	Kg/m^3
Tensile Strength	495000000	N/m^2
Yield Strength	435000000	N/m^2

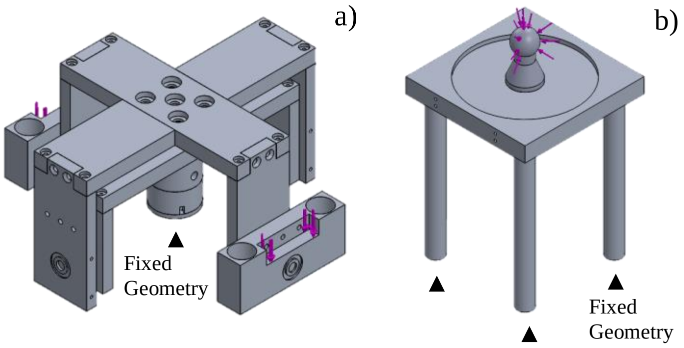


Figure 4. Load applied area a) upper part b) lower part

3.1. Stress formation

Figure 5 shows the stress formation during static state which means large and small frame are inclined 90 degrees from the base. The middle area of the large frame shows only small area at the edge of the frame with a higher stress. The value of von Mises at the red area is 116,590,080 N/m2 which is lower than the yield strength of the aluminium alloy used in the analysis. Therefore, it can be concluded that the frame can perform well during static state without being deformed.

In figure 6, the frame is executing flexion state which is the largest motion in this study. The inclination angle from the base is 65 degree resulting the formation of angle of flexion which is 25 degree. The result shows two edges in the centre of the large frame have highest stress formation. The von Mises value at the red area is 131,382,512 N/m2 which is higher than the value during static state. However, the value still lower than the yield strength of materials used in the simulation.

In figure 7, the result obtained in extension state was almost similar as in flexion state. The difference is the other two edges has higher stress but the value of von Mises not as high as in flexion state which is 127,775,424 N/m2. From both flexion and extension state, it can be seen that only two edges experienced higher force at a time. Although the von Mises value is lower than the yield strength of the material, but it need to be considerate for selection of other materials in the future.

Due to small angle executed in the main frame during abduction and adduction state, there is not much difference can be seen from these states and static state. These three states which are static state, abduction state, and adduction state show same result but only small difference in von Mises value. The abduction state performs an angle of 7 degree while adduction make a 4 degree in other direction. In abduction state, the max von Mises value is 110,270,600 N/m2 and for adduction is 110,082,592 N/m2. Although the value shown is slightly different, but it shows that the larger angle performs by the frame will result in higher stress in the centre of the large frame.

As shown in figure 10, the lower part has the least stress formation in the whole area. There is no red area which mean none of the component will destruct when high load is applied. The highest von Mises value in the analysis is 5,743,119 N/m2 which is the lowest among all main frame result.

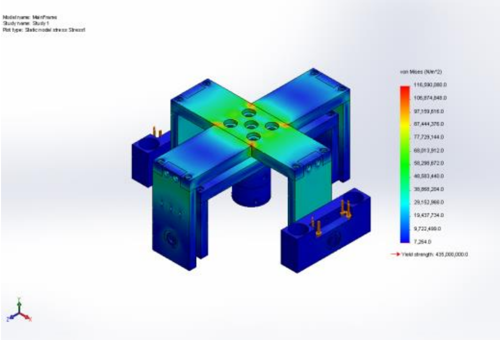


Figure 5. Static state stress formation

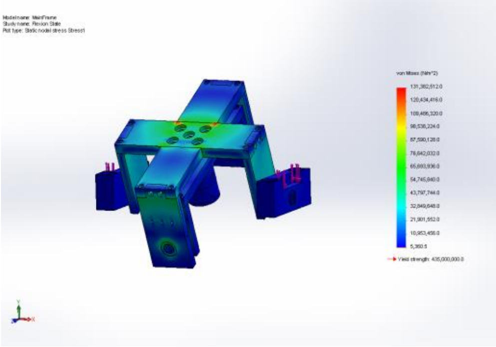


Figure 6. Flexion state stress formation

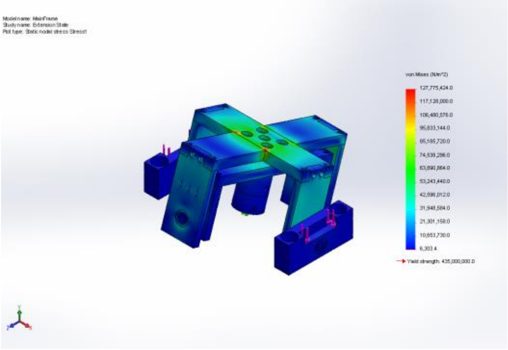


Figure 7. Extension state stress formation

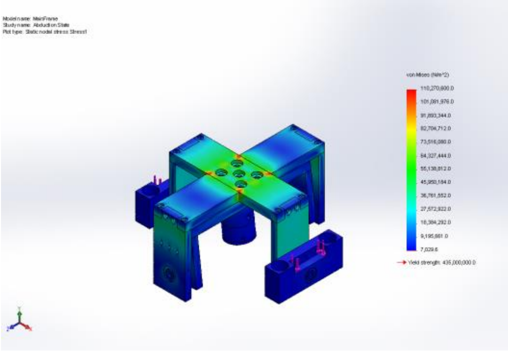


Figure 8. Abduction state stress formation

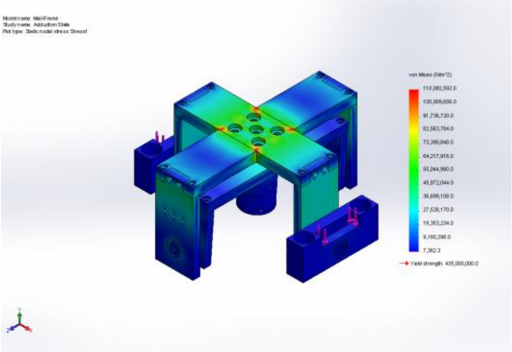


Figure 9. Adduction state stress formation

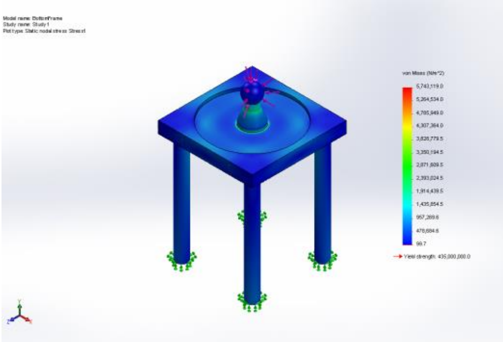


Figure 10. Lower part static state stress formation

3.2. Factor of Safety

From the analysis, safety factor value for all the state are above 2. As shown in figure 11, the FOS value is 2.9 during static state when the load applied. The colour appears in the frame is not important in this simulation because the limit is defined manually from 1 to 100. Reducing the limit will also change the colour appear on the frame analysis.

During the large inclination angle in flexion state, the FOS value is 2.7 which is lower than in static state as illustrated in Figure 12. Similarly, in the stress formation, larger inclination angle will result in high deformation scale thus lowering the FOS value. However, for the extension state, the FOS value was higher than flexion state. The value obtained from extension state in figure 13 is 2.5 which is the lowest FOS value in this analysis. Although the value is the lowest but it still exceeds requirement that needed in this analysis.

For the abduction and adduction state, the FOS value has the same value which is 3.3 as shown in figure 14 and figure 15. Due to slightly different in inclination angel, the result may vary depends on the safety factor distribution.

Furthermore, Figure 16 represents the FOS value for the lower part of the frame. This part is the most stable part based on the simulation analysis. The FOS value give a result of 69 which indicate the part is strong enough to resist any deformation. It is safe to say that lower part is more than capable of receiving load more than 4000 N.

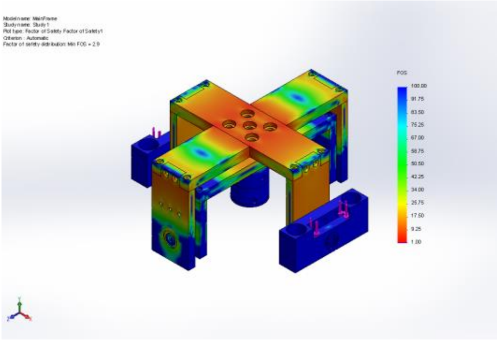


Figure 11. Safety factor value for static state

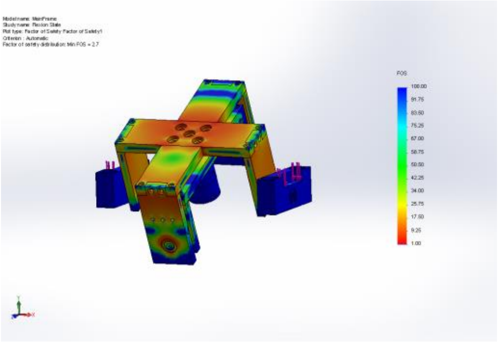


Figure 12. Safety factor value for flexion state

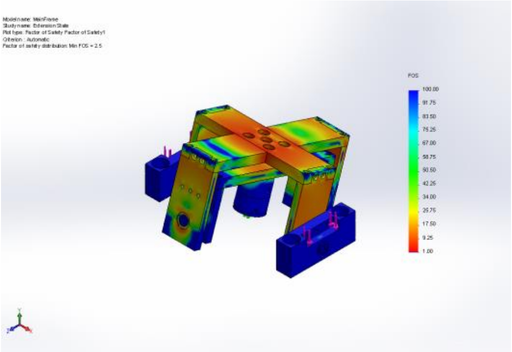


Figure 13. Safety factor value for extension state

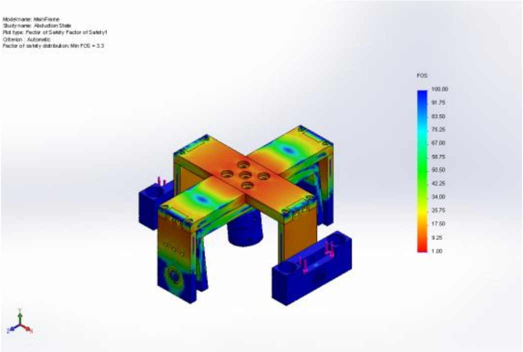


Figure 14. Safety factor value for abduction state

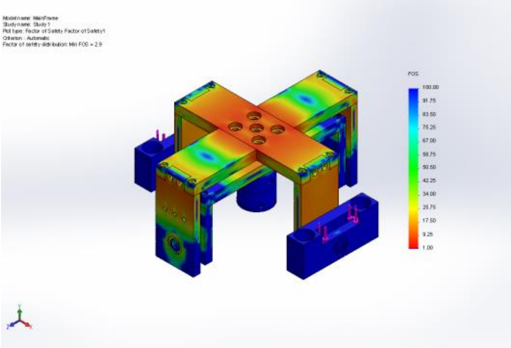


Figure 15. Safety factor value for adduction state

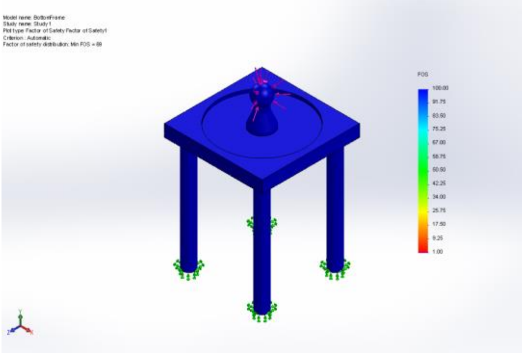


Figure 16. Safety factor value for lower part

The hip simulator is designed to test the hip prosthesis or hip implant in term of wear and dislocation. From the analysis result, it can be shown that the simulator capable to withstand higher load up to 4000 N which is greater than the peak load of previous simulator. Further study can be executed for testing heavier load and enhance the stability of the structure.

4. Conclusion and recommendation

The finite element analysis of main frame 2-station hip simulator has been successfully implemented using SolidWorks software. Based on the result of stress formation and factor of safety value, it can be concluded that selected material which is aluminium alloys 7050-T73510 is a suitable material for the design. Besides, no further major modification need to be done. The result shows with the load applied up to 4000 N, deformation scale of the parts is low and the von Mises value does not exceed yield strength of the selected material. Factor of safety showing value results more than 2 for all the parts which indicate that the parts will not permanently deformed. These results will provide information and used as reference for determining the structure of hip simulator in further research.

5. Acknowledgments

The authors would like to thank the Universiti Teknologi Malaysia (UTM) and Ministry of Education (MoE) Malaysia for financial support through Research University Grant (GUP) with Vote No.13H23.

6. References

[1] Kiguchi K, Yamashita A, Sasaki M, Ueno M, Kobayashi T, Mawatari M and Hotokebuchi T 2008 *Proc. of Int. Conf. on Control, Automation and Systems (Seoul)* (South Korea: IEEE) p 1942–1945
[2] Bragdon C R, Jasty M, Muratoglu O K, O'Connor D O and Harris W H 2003 *Journal of Arthroplasty* **18** 553–561
[3] Trommer R M and Maru M M 2016 *Revista Brasileira de Ortopedia (English Edition)* **52** 251-259
[4] Galanis N I and Manolakos D E 2011 *Proc. of the World Congress on Engineering (London)* vol 3, ed S. I. Ao and Len Gelman *et al* (U. K: Newswood Limited) p 2–7
[5] Affatato S, Leardini W and Zavalloni M 2006 *11th BIOLOX Symposium Proc.* (Rome) vol 11 (Darmstadt: Steinkopff) p 171–180
[6] Saikko V and Calonius O 2002 *Journal of Biomechanics* **35** 455–464
[7] Calonius O and Saikko V 2002 *Journal of Biomechanics* **35** 1439–1450