ANALYSIS OF POLYMETHYLMETHACRYLATE AS BONE SUBSTITUTE OF FRONTAL HUMAN SKULL VIA FINITE ELEMENT ANALYSIS

Shahrul Hisyam Marwan[1], Abdul Rahim Bahari[1], Mohd Faizul Idham[1], Haizuan Abd. Rahman[1], Mohamad Daniel Benjamin Mohd Radzuan[2]

[1]Faculty of Mechanical Engineering, Universiti Teknologi MARA, Bukit Besi, Terengganu, Malaysia
[2]Faculty of Chemical Engineering, Universiti Teknologi MARA, Bukit Besi, Terengganu, Malaysia
E-Mail: shisyam18@gmail.com

ABSTRACT

A study has been conduct to verify the compatibility of the biomaterials which is Polymethylmethacrylate (PMMA) by developing a frontal human head impact test using finite element analysis (FEA). This study is to compare the results between simulation and experimental result conducted by previous study. To conduct this study, the process involve is to develop finite element model of spherical skull in SOLIDWORKS and to analyze the data analysis using ANSYS Workbench. It is to study the impact conditions of human skull during a human head impact on the resulting of total deformation. Three finite element model of spherical skull are created using SOLIDWORKS, which the three of them is distinguished by their thickness which resembles the thickness of real frontal human skull. Then, all of the models are exported to the software called ANSYS Workbench to create a simulation of a frontal human head impact test. The result of the simulation is total deformation is then analyze by calculating the percentage error and percentage difference between the simulation result and the experimental result by previous study. The average percentage error between the simulations of spherical skull models with previous experimental result is 12.79%, while the lowest percentage error is around 10.91%. The average percentage difference for the simulation result using PMMA with the previous experimental result is around 4.74%, and the lowest percentage difference is around 3.00%. In conclusion, the result shows the PMMA is suitable as bone substitution for frontal human skull.

Keywords: ANSYS, PMMA, solidworks, total deformation, human skull thickness, bone substitution.

1.   INTRODUCTION

Nowadays, the finite element modelling and simulation is a significant tool for studying injuries of the human head subject to impact loadings compared to the experimental method. It is to verify better materials to use as bone substitute. As finite element models of the head observed wider applications in a diversity of fields, verifying new materials to substitute bone is now a big concern in surgical procedure. Presently, lots of experimenter tries to mimic this real situation by conducting head impact test. Investigation of the relation between mechanical load and the resulting injury can lead to the development of brain injury criteria. Hence, finite element modeling is a powerful technique through which impact-tolerance can be studied. The finite element analysis (FEA) is a method that reconstructs and evaluates the stress, strain and deformation of structures and it's also an illustration of a structure that is readily resolved by mathematical analysis as a series of subdivisions [1]. By using a dynamic analysis program called ANSYS Workbench, it is possible to verify the suitable biomaterial via simulation mimicking the real situation of the accidents in order for us to further to study.

Majority of car accident these days impacted on head which is the frontal position. Hence new biomaterials have emerged to replace the fracture bone and repair the bone tissue. In order to fix the problems that involving bones or joints, a surgical procedure is introduced that called as bone grafting. Ceramics and polymer with desirable properties such as biocompatibility and osteoconductivity are used for bone substitute these days

but the only missing pieces in these bone substitute is the ability to bear loads [2]. According to [3] sintered Hydroxyapatite bio-ceramics cannot be used for heavy load-bearing bones due to low mechanical properties [4]. Hence, a new biomaterial has been proposed which is Polymethylmethacrylate (PMMA) that has the mechanical properties which resemble a real frontal skull. Biomaterials that been used as bone substitute must be biocompatible to the human condition to avoid a chronic immune response by the host and also should be osteoinductive to allow the formation of new bone tissue and adequate bone ingrowths [2]. In order to verify the compatibility of this biomaterial, a dynamic analysis program is used which is ANSYS Workbench. According to [5], the assessment of accidents statistics shows that beyond 30% of all vehicular traumas are related to head injuries [6]. One of the purposes of this paper is to discover a suitable bone substitute that mimicking the real human skull. Nowadays there are so many biomaterials that have been used as bone substitutes but none to replace the frontal human skull. After reviewing lots paper, it is found that PMMA has mechanical properties that resemble the frontal human skull. With a validated finite element of spherical skull model, it is possible to verify the biomaterial if it's suitable to use for bone substitution of frontal human skull.

A study has been conduct to verify the compatibility of the biomaterials which is PMMA by developing a frontal human head impact test using FEA. This study is to compare the results between simulation and experimental result conducted by [7]. To conduct this

www.arpnjournals.com

study, the process involve is to develop finite element model of spherical skull in SOLIDWORKS and to analyze the data analysis using ANSYS Workbench which to study the impact conditions of human skull during a human head impact on the resulting of total deformation. Three finite element model of spherical skull are created using SOLIDWORKS which the three of them is distinguished by their thickness which resembles the thickness of real frontal human skull. Then all of the models are exported to the software called ANSYS Workbench to create a simulation of a frontal human head impact test. The result of the simulation is total deformation is then analyze by calculating the percentage error and percentage difference between the simulation result and the experimental result by [7]. The total deformation of each tissue can be easily observed by using Finite Element Model [8, 9]. The average percentage error between the simulations of spherical skull models with Delye's experimental result is 12.79% while the lowest percentage error is around 10.91%. The average percentage difference for the simulation result using PMMA with the Delye's experimental result is around 4.74% and the lowest percentage difference is around 3.00%. In conclusion, the result shows the PMMA is suitable as bone substitution for frontal human skull.

2. METHODOLOGY

Finite element (FE) model

In this study, three FE model of spherical skull are created by SOLIDWORKS 2014 using the specification data that have been extracted from [7,10]. The spherical skull model is assumed as the human skull as shown in Figure-1. Each of the models is created with different thickness and diameter and each of them are labelled with the name of Skull 1, Skull 2 and Skull 3 respectively. The models created consist of 3 bodies which is the spherical skull model, the impactor and the block that is attached to the impactor. The mechanical properties of real frontal human skull are applied to the spherical skull models that have been created.
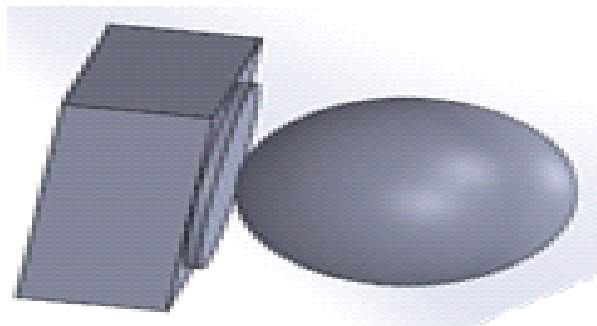

Figure-1. Spherical skull model with impactor.

The impactor and block created for three models use the same materials which is the impactor is made up of aluminum plate and the block is made up of steel. The

impactor is attached to the block to support the impactor when being impacted by the spherical skull model. Both impactor and block give the weight of 49.56 kg in totals.

Validation of FE model

Each of the spherical skull models need to be validated first before the simulation for PMMA can be conduct. The explicit dynamic analysis is used in this study and all the mechanical properties for real frontal human skull is then being applied in the engineering. The models that have been created before in SOLIDWORKS will then import to the ANSYS Workbench. The exported model then must be generated first in order the model from SOLIDWORKS becomes a 3D models. The boundary conditions for all the geometry for the models must be applied and then the models are meshed. The initial conditions of the models must be applied such as the velocity and the fixed support as shown in Figure-2.
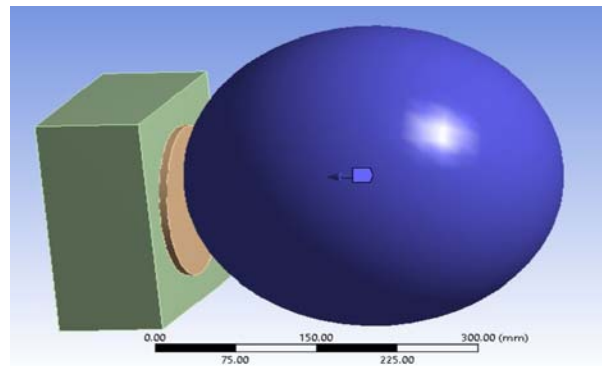

Figure-2. Initial conditions of the FE model.

Finite element analysis

After the spherical skull models are validated, the mechanical properties of the frontal human skulls are then replaced using the mechanical properties of PMMA. The mechanical properties involved are Young Modulus, Poisson's Ratio, and the density of PMMA which resembles the mechanical properties of real frontal human skull. The result of the simulation which is the total deformation is taken and analyses. The result of the simulation is then being compared with [7] and the graph of the different between the simulation result and the experimental result is constructed. The percentage error between the result is then calculated in order to identify the approximately error.

3. RESULTS AND DISCUSSION

The analyses have been done on the spherical skull models are total deformation using ANSYS Workbench which to study the displacement resulted from the impact between the spherical skulls with the impactor. Based on the analysis, the total deformation for each of the spherical skull models that been replaced with PMMA is almost the same with the total deformation result conducted by [7]. The impact velocities differed from 5.19 m/s, 5.32 m/s, and 6.97 m/s. The impactor condition is

www.arpnjournals.com

generated in the shape of cylindrical that is attached to the block with different velocities and a mass of 49.56 kg.

Table-1 consists of thickness, diameter, velocities, impactor weight, Delye's experiment displacement, simulation's displacement for real frontal human skull, and also percentage error are created for each of the skull identification number (ID). Table-2 consists as said before but with a difference simulation result which is simulation result for PMMA. The table is constructed in order to study their respective total deformation. The percentage error is calculated to compare an estimate to an exact value. The percentage error gives the difference between the approximate and exact values as a percentage of the exact value and can help to see how close the guess or estimate is to a real value. Eqn. (1) is the formula to calculate percentage error for each of the skull ID involve. Eqn. (2) is the percentage difference between both results. Percentage difference is calculated in order to know how much the value is differ. Eqn. (1) and Eqn. (2) are used to calculate the percentage error and percentage difference respectively.

$$\%_{error} = \frac{|Simulation\ Result - Experiment\ Result|}{|Experiment\ Result|} \times 100\% \quad (1)$$

$$\%_{Difference} = \frac{Experiment\ Result - Simulation\ Result}{\left(\frac{Experiment\ Result + Simulation\ Result}{2}\right)} \times 100\% \quad (2)$$

Table-1. Result of spherical skull models validation.

Skull ID	1	2	3
Diameter (mm)	177.3	164.4	184.2
Thickness (mm)	6.59	4.88	8.67
Impactor weight (kg)	49.56		
Velocity (m/s)	5.19	5.32	6.97
Delye's experiment displacement (mm)	3.80	4.40	4.40
Simulation's displacement for real frontal human skull (mm)	3.24	3.84	3.92
Percentage error (%)	14.74	12.73	10.91

Table-2. Result of PMMA as bone substitution.

Skull ID	1	2	3
Delye's experiment displacement (mm)	3.80	4.40	4.40
Simulation's displacement for PMMA (mm)	3.50	4.27	4.27
Percentage difference (%)	8.22	3.00	3.00

A graph is created to show the total deformation from the simulation result and the experiment result. Figure-3 shows the simulation result that is almost the same with the [7] experimental result. This does not seem to be as a result of a wrong value in these mechanical parameters but to a lack of the model that seems to need the introduction of elements with a damping and/or retaining action for the brain tissues. Modelization of the cerebrospinal fluid (CSF) is one of the main problems to

the differences result. A structural analysis without fluid elements cannot properly simulate the fluid damping and the fluid-dynamic migration of the CSF in different areas during impact although the very short duration of the phenomenon [11].
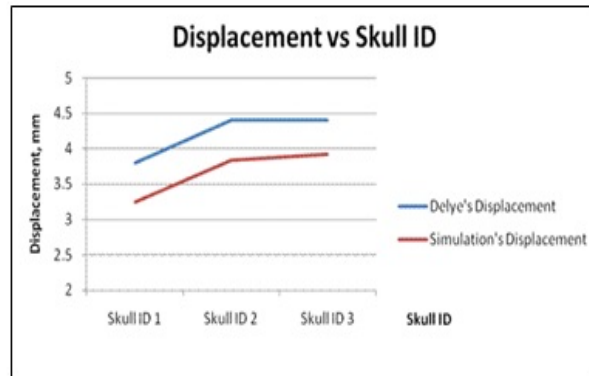

Figure-3. Graph of displacement against skull ID for spherical skull model.

Figure-4 shows the graph of the displacement between Delye's displacement and simulation's displacement of PMMA. The difference between them is quite smaller even though the mechanical properties of PMMA are approximately the same to the real frontal human skull. From the graph, it could be conclude that PMMA have a higher potential to substitute the frontal human skull. The mechanical properties of PMMA also can easily adapt to human condition.
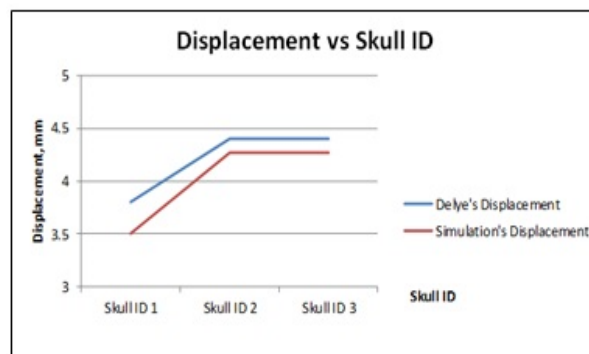

Figure-4. Graph of displacement against skull ID for PMMA as bone substitution.

4. CONCLUSIONS

In conclusion, objective that have been achieves are discovering that PMMA as suitable bone substitute for frontal human skull that mimicking the real frontal human skull, verifying PMMA for bone substitute via FEA, and differentiate the result from analysis which is total deformation with previous study experimental result. The total deformation result of frontal skull impacted for each model are compared and it is approximately resemble with the result that previous study experimental discovered.

www.arpnjournals.com

From the results that have been acquired, the conclusion that could be make is the models that have been created is a successful and is consider suitable for the simulation to be done. According to the results acquired based on the simulation, the spherical skull model is validated by receiving an average of 12.79% of the percentage of error which is moderate. The highest percentage error for validated frontal human skull model is 14.74% and the lowest value is 10.91%. For the model of frontal human skull using the PMMA properties, the average percentage difference is 4.74% which is small. In this simulation, two of the models magnificently achieved low percentage of error which both of the value is 3.00%. After taken lots of consideration, the mechanical properties of PMMA have been concluded that it have slight advantages than the real frontal human skull subjected to three parameters which is in terms of Young Modulus, Poisson's Ratio, and also density. As for the conclusion, the results acquired by the simulation can be considered as suitable to use as a bone substitute even though the percentage difference between the simulation and from previous study experimental analysis is small.

REFERENCES

[1] F. B. Prado, A. C. Rossi, A. R. Freire and P. H. F. Caria, 2014. The application of finite element analysis in the skull biomechanics and dentistry. Indian Journal of Dental Research. 25(3): 390-397.

[2] Lee H. Y. 2012. Composite bone substitutes prepared by two methods. Master thesis, University of California, USA.

[3] H. J. A. Van Dijk, N. Hattu and K. Prijs. 1981. Preparation, microstructure and mechanical properties of dense polycrystalline hydroxy apatite. Journal of Materials Science. 16(6): 1592-1598.

[4] H. R. Le, S. Qu, R. E. Mackay and R. Rothwell. 2012. Fabrication and mechanical properties of chitosan composite membrane containing hydroxyapatite particles. Journal of Advanced Ceramics. 1(1): 66-71.

[5] Allsop D. and Kennett K. 2002. Skull and facial bone trauma. In: Accidental Injury. A. M. Nahum and J. W. Melvin (Eds.). pp. 254-276.

[6] Z. Asgharpour, D. Baumgartner, R. Willinger, M. Graw and S. Peldschus. 2014. The validation and application of a finite element human head model for frontal skull fracture analysis. Journal of the Mechanical Behavior of Biomedical Materials. 33: 16-23.

[7] H. Delye, P. Verschueren, B. Depreitere, I. Verpoest, D. Berckmans, J. Vander Sloten, G. Van Der Perre and J. Goffin. 2007. Biomechanics of frontal skull fracture. Journal of Neurotrauma. 24(10): 1576-1586.

[8] S. H. Marwan, A. H. Abdullah, J. Mahmud and H. Rashid. 2013. Dynamic analysis of frontal human skull using finite element Simulation. Advanced Materials Research. 647: 418-423.

[9] Jiroušek O., Jíra J., Jírová J. and Micka M. 2005. Finite element model of human skull used for head injury criteria assessment. In: IUTAM Symposium on Impact Biomechanics: From Fundamental Insights to Applications. M. D. Gilchrist (Ed.). pp. 459-467.

[10] Oñate E. 2013. Structural analysis with the finite element method linear statics-Volume 2: Beams, Plates and Shells. 1st Ed. Springer Science & Business Media. Heidelberg, Germany.

[11] Belingardi G., Chiandussi G. and Gaviglio I. 2005. Development and validation of a new finite element model of human head. In: 19th International Technical Conference of the Enhanced Safety of Vehicle. 35: 1-9.