Journal of Fundamental and Applied Sciences

ISSN 1112-9867

Available online at http://www.jfas.info

Research Article

Special Issue

COMPARISON BETWEEN CARBON FIBRE REINFORCED PLASTIC, ALUMINIUM ALLOY AND ALLOY STEEL FOR CAR HOOD APPLICATION FOR IMPACT TEST BY USING FINITE ELEMENT ANALYSIS

M. Mohamed[1,*], M. F. M. Din[2], F. R. Hashim[2], M. H. M. Amini[1], M. N. Masri[1], M. B. A. Bakar[1], M. K. A. A. Razab[3] and Z. I. Rizman[4]

[1]Faculty of Bio-Engineering and Technology, Universiti Malaysia Kelantan (UMK), Jeli Campus, 17600 Jeli, Kelantan, Malaysia

[2]Department of Electrical and Electronic, Faculty of Engineering, National Defense University of Malaysia, Sungai Besi Camp, 57000 Kuala Lumpur, Malaysia

[3]School of Health Science Universiti Sains Malaysia, Health Campus, 16150 Kota Bharu, Kelantan, Malaysia

[4]Faculty of Electrical Engineering, Universiti Teknologi MARA, 23000 Dungun, Terengganu, Malaysia

Published online: 01 February 2018

ABSTRACT

This paper presents about importance to know the best material need to be applied using FEA in case of crashes in order for the hood to be rigid and enable safety. The increase of strength of car hood can increase the safety of car engine and car drivers. The SDA was investigated numerically using FEA. It simulates the impact test for car hood using SolidWorks simulation software. 3 different materials of car hood (Al 6061 T6 (SS), Steel AISI 1045 and CFRP-Sandwich composite) were examined by 4 different forces (870 N, 1370 N, 1870 N and 2370 N) that impacting the car hood.

Author Correspondence, e-mail: mazlan.m@umk.edu.my

doi: http://dx.doi.org/10.4314/jfas.v10i2s.34

The simulation result using different material of maximum stress and maximum displacement was successfully compared. CFRP-Sandwich composite has higher value of maximum displacement compare to Steel AISI 1045 and Al 6061-T6 (SS). CFRP-Sandwich composite has high flexibility in material properties. CFRP-Sandwich and Steel AISI 1045 almost have same value of stress. Both materials have high yield strength performance compared to Al 6061-T6 (SS).

Keywords: finite element analysis; carbon fiber reinforcement; Solidworks; car hood.

1. INTRODUCTION

Nowadays, the automotive industry in Malaysia is getting better and wider day by day. Many improvements have been done to increase the quality of the automotive products and at the same time trying to preserve the environment. A significant effort made by the automotive and component manufacturers to reduce aerodynamic drag, noise and vibration. Automotive designer rely on aerodynamic principle to create improvement in the power and handling of vehicle at high speeds. The design can reduce air resistance, or aerodynamics drag, Low drag coefficient make the vehicle enable to move easily through the surrounding viscous air with minimum resistance [1].

The hood of a car today gives the impression of being a simple solution. A car hood is the metal part that covers the engine of an automobile. Other than that, car hood also known as car bonnet which is a British term and it is means cloth used as headdress. With increased complexity on hinges and lock mechanisms, the hood opening mechanism is not that simple. Opening and closing the hood also demands a gap between the hood and the body wide enough to avoid clash [2].

It can also be considered as another door hood of vehicle. It is getaway to get access to the front of engine compartment of vehicle engine. It consists of inner and outer panels. For inner panel serves as abdominal support panel off, so it would not be easy to flex and get dented. On the other hand, outer panel is designed to meet sheet metal and tenders, edge contour matches the cowl. Furthermore, hoods come in designs and styles. Some automotive hoods are covered with sound-absorbing material to minimize the sound escape the engine bay.

Another design that is found in automotive hood is spoon. The function of spoon is allowing the engine bay spoon to inhale and exhale. It is also to release the hot under hood air, which also helps the performance of the engine. Hood shaped and contoured to perfection to radiate the presence of the day, it also serves as a protective cover for the engine to prevent the intrusion of water and debris. Hoods that have high grade and durable construction effectively shield the rain, snow, hail or debris that can reach into car engine and disrupt the operation.

By only looking on the vehicles front propagation can maintain a good quality engine and ensure the excellent performance in keeping away from damage. Moreover, in automotive industry, hood usually fabricated from steel, glass, carbon fibre and aluminium. The most popular material that had been used is aluminium. It is because it can provide pliability, strength and easy to handling. Materials that used in fabricating hoods are Bake Hardening steel, aluminium alloy, magnesium alloy, titanium alloy and Carbon Fibre Reinforced Plastic (CFRP) [3].

Recently, increased numbers of pedestrian injuries are resulting from road accidents due to the increasing vehicles. To overcome this issue, safety measurement, design, material of engine hood has been a key point of automotive industry. The most common material that used was steel, aluminum and composite material [4]. In automotive industry has been growing interest in the use of naturally sourced fibres for use in composite design and manufacture. A structural design on automobile hood using natural flax fibre composited was performed [5].

1.1. Material Used In Manufacturing Automotive Hood

Carbon Fibre Reinforced Plastic (CFRP), Carbon Fibre Reinforced Polymer (CRP) and Carbon Fibre Reinforced Thermoplastic (CFRTP) contain carbon fibre which is an extremely strong and light fibre-reinforced plastic. CFRPs are commonly used herever high strength-to-weight and rigidly are required such as automotive, aerospace, civil engineering, sports goods and an increasing number of other consumer and technical applications. CFRPs also can be expensive. The vital thing in composite component is thermoset. Thermoset is the binder of polymer, for example, epoxy resin. Others thermoset or thermoplastic polymer such as polyester, vinyl ester or nylon are sometimes used.

The composite may contain other fibres, such as aramid, aluminium,

ultra-high-molecular-weight polyethylene (UHMWPE) or glass fibres. There are type of additives that introduced to the binding matrix can affect the properties of the final CFRP product. The additives that can be used are silica, rubber and carbon nanotubes. CFRPs are composite materials that consist of two parts; a matrix and reinforcement. In CFRP, the reinforcement is carbon fibre which provides the strength. The matrix is usually a polymer resin such as epoxy to bind the reinforcement together. The reinforcement will give the CFRP its strength and rigidity. It can be measured by stress and elastic modulus respectively.

Typical epoxy-based CFRPs exhibit virtually no plasticity, with less than 0.5% strain to failure. Although, CFRPs with epoxy have high strength and elastic modulus, the brittle fracture mechanics present unique challenges to engineers in failure detection since failure occurs catastrophically.

Environmental effects such as temperature and humidity can have profound effect on the polymer-based composites, including most CFRPs. While, CFRPs demonstrate excellent corrosion resistance and the effect of moisture at wide ranges of temperature can lead to degradation of mechanical properties of CFRPs. Carbon fibres are not affected by the moisture diffusing into the material. Therefore, the carbon fibre can cause galvanic corrosion when carbon fibre reinforced plastics are attached to aluminium.

For automotive engineering, CFRPs are extensively used in high end automobile racing. The high cost of carbon fibre is reduced by the material's unsurpassed strength-to-weight ratio and low weight is essential for high performance automobile racing. Recently, several mainstream vehicle manufacturers have started to use CFRP in everyday road cars. Use of material has been more readily adopted by low-volume manufacturers, who used it primarily for creating body-panels for some of their high-end cars. Due to its increased strength and decreased weight compared with the glass-reinforced polymer, they used for the majority of their products.

Carbon fibre can be used to reduce weight and hence the size of its frame. A preference for carbon fibre can also reduce the amount of water and electricity used in manufacturing. Effective requirements management plays an important role when it comes to support of product development teams in the automotive industry. A precise positioning of new cars in

the market is based on features and characteristics described as requirements as well as on costs and profits [6].

Study concludes that in most of the cases, an accident occurs with the pedestrian and work has been done on the design of front head and near head panel. For safety of the bonnet, work should be done on the stiffness of the material. Hybrid materials can be suggested for better performance of the bonnet. In most of the cases steel and aluminium are used [7]. Carbon fibre reinforced thermoplastic are increasingly used for high performance composites, particularly for the automotive industry. A study had focuses on the development of short carbon fibre reinforced polypropylene for use as car bonnet in replacement for steel [8].

In fibre reinforced polymer composite, it has high cost of carbon fibre, limited weight reduction potential of glass fibre, long cycle times for many processes; difficulty incorporating structure at many length scales into processing and design models. Other composite that is used to design car hood is graphene-based polymer composite. This is because the material will provide benefits such as improved strength, flame behaviour and superior durability [9].

Engineered composites composed of un-oriented, discontinuous are continuous low-cost fibres such as glass and a commodity thermoplastic matrix like polypropylene or a thermoset matrix such as polyester are already well-established in the automotive industry. Carbon fibre composites have a number of convincing properties in their favour such as high strength and durability. But most significantly, as far as the global automotive industry is concerned, they are 50% lighter than steel [10].

1.2. Solid Modelling Using SolidWorks

SolidWorks is a solid modelling computer-aided design (CAD) and computer-aided engineering (CAE) computer program that runs on Microsoft Windows. It is published by Dassault Systemes. SolidWorks is a solid modeller and utilizes a parametric feature-based approach to create models and assemblies. Parameters refer to constraints whose values determine the shape or geometry of the model or assembly. Parameters can be either numeric parameters such as line lengths or circle diameters or geometric parameters such as tangent, parallel, concentric, horizontal or vertical.

Numeric parameters can be associated with each other through the use of a relation, which allows them to capture design intent. Computer Aided Design (CAD) is revolution in today's competitive industry. In mechanical industry, design process is very time consuming because it is iterative and requires experienced peoples. CAD tool can be used for various applications in mechanical engineering resulting less time in design and better productivity and quality. These are many Computer Aided Design (CAD) packages available in market for design which demands modelling skills [11].

SolidWorks can analyze the linear studies which include buckling, drop test, dynamic analysis, fatigue, frequency, harmonic analysis, optimization, pressure vessel design and random vibration, static, thermal and transient thermal [12-16]. Solid simulation is usually for test the properties of the materials which apply to the certain industry, while flow simulation for testing the flow system. SolidWorks also used in robot modelling which used to scrutinize the theory and its motion simulation with generalized to a 3-R robot [17].

1.3. Finite Element Analysis Software

Nowadays, analytical methods offer rough estimates on product performance in many design application. By using finite element analysis software, it can evaluate performance of the design decision making. Finite element analysis tools can be used by design engineer to solve the problems that beyond hand calculation and analytical methods [18].

Finite element analysis is a Solidworks simulation. The function of the methods is to analyze structural performance. It is also a computerizes method for predicting how a product reacts to the real-world forces, vibration, stress, heat, fluid flow and others physical effects.

Finite Element Analysis (FEA) is a numerical technique to find approximate solution of partial differential equation. Widely used in the field of mechanical system design which can explained the behavior of individual element with simple equation, and this set equation described the behavior of the whole structure when combine with big set of equation [19].

A study that has been carried out by [20], the modelling and simulation of the stop valves are created numerically by using Solidwork. Three-dimensional numerical simulations were conducted to observe the flow patterns and to measure valve flow coefficient and flow fluctuations when stop valve with different flow rate and uniform incoming velocity were

used in a valve system.

Finite element method works according to discretize the continuum, select interpolation functions, find the element properties, assemble the element equation, solve the global equation system and compute additional results [21].

Application that will develop the great system of material such as structure of agricultural machinery like harvester coffee machining, automotive, corrosion and welding field [22] and dentistry in medical [23]. The application in automotive currently is dry friction clutch, 4-cylinder diesel crankshaft [24] and design sheet metal [25] in comparison with experimental measurement result and constant sliding speed demonstrated in this analysis.

Finite Element Method (FEM) is a numerical method such as finite difference method but more general and powerful in its application to real-world problems. By using FEM for the machine structure in analysis, it was possible to generate values of stresses and displacements for both situations which allowed the choice of suitable design for structural components of a machine coffee harvester [26].

In the present work a friction clutch assembly was designed and a model of the same was created in Solid Works Office Premium Software. It consist of three parts viz. clutch plate, pressure plate and diaphragm spring. Finite element analysis was performed in ANSYS software. The finite element analysis was carried out in three steps which are Pre-processing, Solving and Post processing. The plots for Equivalent von-Misses stress, total deformation and stress tool (factor of safety) were calculated and analyzed. The finite element analysis showed that the designed friction clutch assembly is safe [27].

Finite element model used to investigate the transient thermo elastic process, spatial discretization and modal superposition is presented. Constant sliding speed was performed in this analysis. Therefore, finite element analysis also used to find the thermal behavior and comparison with experimental measurement results [28].

1.4. Impact Test for Automotive Hood

More than 270,000 pedestrians lose their lives on the world's roads each year accounting for 22% of the total 1.24 million road traffic deaths. Despite the magnitude of the problem, most attempts at reducing pedestrian deaths have focused solely on education and traffic regulation.

Crash engineers have begun to use design principles that have proved successful in protecting car occupants to develop vehicle design concepts that reduce the likelihood of injuries to pedestrians in the event of a car-pedestrian crash. These involve redesigning the bumper, hood (bonnet) and the windshield and pillar to be energy absorbing (softer) without compromising the structural integrity of the car.

An impact test is a technique for determining the behavior of material subjected to shock loading in bending, tension and torsion. This test is designed to determine how a specimen of a known material will respond to a suddenly applied stress. The test ascertains whether the material is tough or brittle. It is mostly used to test the toughness of metals, but similar tests are used for polymers, ceramics and composites.

Metal industry sectors that use the impact test include oil and gas, aerospace, power generation, automotive and nuclear. The impact test is a method for evaluating the toughness, impact strength and notch sensitivity of engineering materials.

Engineers test the ability of a material to withstand impact to predict its behavior under actual conditions. Many materials fail suddenly under impact, at flaws/cracks or notches. The most common impact tests use a swinging pendulum to strike a notched bar; heights before and after impact are used to compute the energy required to fracture the bar. In the Charpy test, the test piece is held horizontally between two vertical bars. In the Izod test, the specimen stands erect like a fence post.

The notched test specimen is broken by the impact of a heavy pendulum or hammer, falling at a predetermined velocity through a fixed distance. The quantity usually measured is the energy absorbed in breaking the specimen in a single blow, as in the Charpy impact test and Izod impact test.

Impact tests are also performed by subjecting specimens to multiple blows of increasing intensity, as in the drop ball impact test and the repeated blow impact test. Impact resilience and Scleroscope hardness are determined in non-destructive impact tests. A human body finite element model called Total Human Model for Safety (THUMS) is developed to study the detail injury occurrence mechanism instead of crash test dummies. First, modelling of lower extremity in (THUMS) is validated against cadavers of shearing and bending loads by

comparing the dynamic response of a pedestrian's lower extremity due to loading from a car bumper. Secondly, (THUMS) is validated against the body kinematics of lower extremity impact [29].

2. METHODOLOGY

The method that used in this project is SolidWorks simulation with finite element analysis. Function of Solidworks software is to designs model quickly and precisely. SolidWorks designs are defined by 3D design based on the model or component that had been selected. This method is a common numerical methods used for simulations in mechanical engineering. By using finite element analysis (FEA), it can solve mathematically difficult and time consuming operation.

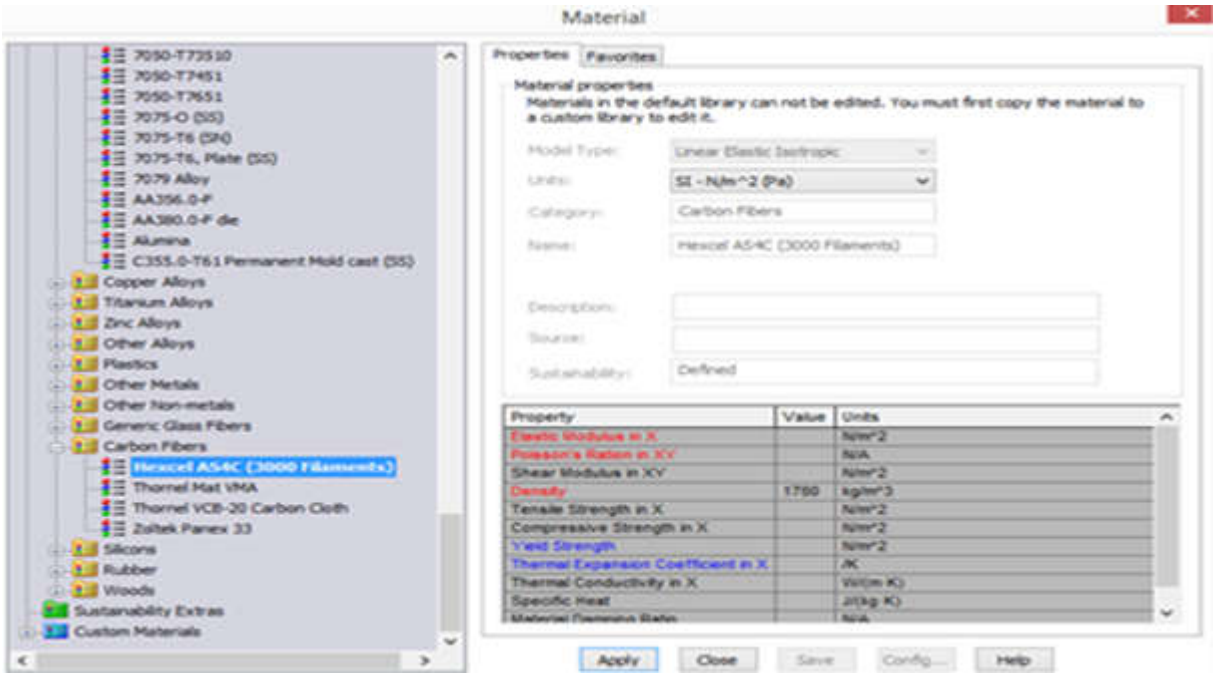
Therefore, SolidWorks simulation is software that can allows the use of FEA. The finite element method is a numerical method for solving problems of engineering and mathematical physics. Typical problem areas of interest in engineering and mathematical physics that are solvable by use of finite element method include structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential. For problems involving complicated geometries, loading, and material properties, it is generally not possible to obtain analytical mathematical solutions.

Analytical solutions are those given by a mathematical expression that yields the values of the desired unknown quantities at any location in a body (here total structure of physical system of interest) and are thus valid for an infinite number of locations in the body. These analytical solutions generally require the solution of ordinary or partial differential equations which because of the complicated geometries, loading and material properties are not usually obtainable [30].

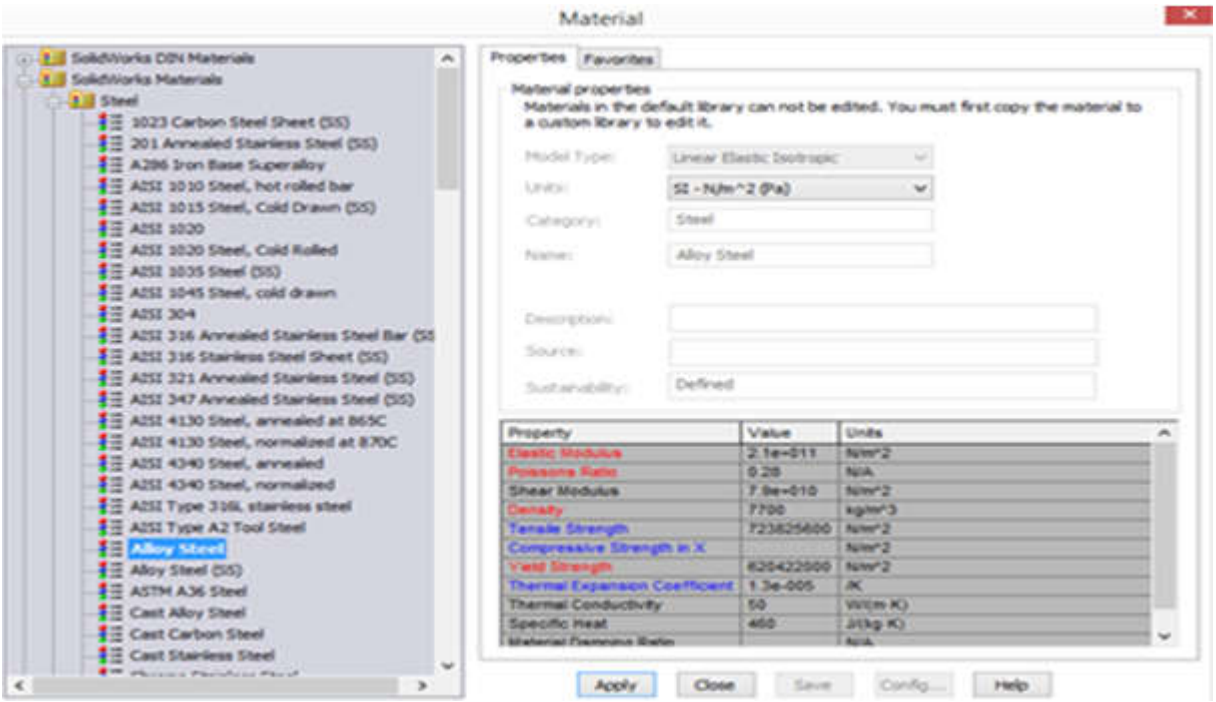
The model design in this project is a car hood in automotive application. The dimension of the car hood that had been taken is from the real car hood of Perodua Kancil. FEA can analyze structural properties of the car hood and it also provides motion analysis. It is used for accurate simulation and analysis of moving assemblies. The elements of kinetic studies and bonds will be combining in the analysis in the motion calculation. Using Finite Element

Analysis which widely used in automotive sectors to investigate and identify the suitable materials for the car hood part.

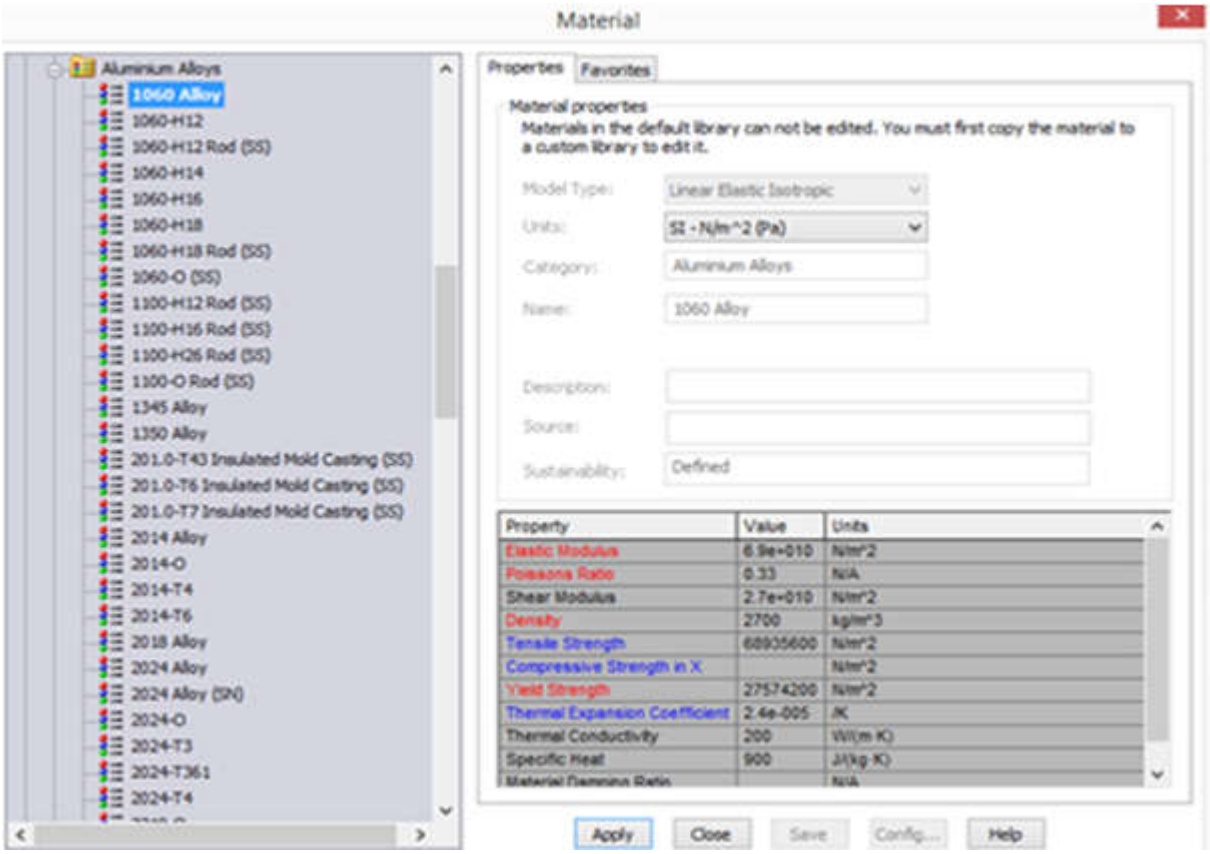
In this study, the parameters that will be used are different types of materials that are used for car hood modelling and the forces that have been set for the impact test on the car hood model. The materials that have been chosen are aluminium alloy, carbon fibre and steel.



(a)



(b)



(c)

Fig.1. Material used for the simulation carbon fibre, b) alloy steel and c) aluminium alloy

3. RESULTS AND DISCUSSION

3.1. Analysis of Impact Test on Car Hood

Crash test are frequently used to help evaluate car safety. Different car safety programs and organizations specify how such test should be performed, what factors should be investigated and how car safety should be assessed. Impact test are not only performed when the car design is completed and prototype is ready, but also throughout the whole development and validation process. Virtual crash experiment, using finite element analysis can help reduce the number of full-scale car test. Result from these experiments can be used to predict real car behaviour, interactions between a car and its occupants or deformation during collision. Nowadays, finite element analysis is the most popular analytical tools in modelling the impact test performance. The major advantage of finite element analysis model is its capability to

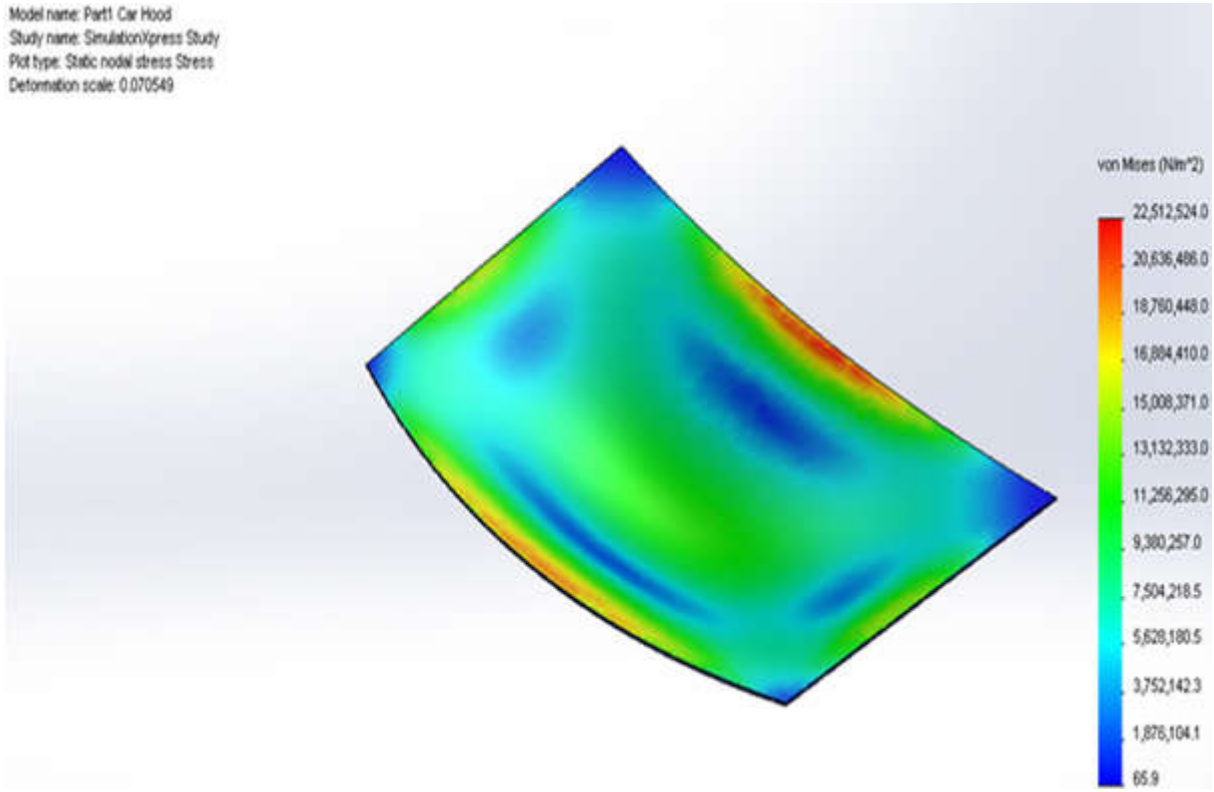
represent geometrical and material details of the structure.

Static analysis which is a test of the strength of car engine hood was conducted by applying forces at selected point in various components of the engine hood for determining resulting displacement and stresses. This can also be divided into static global and closure force finite element analysis. For stress results are compared with elastic limit of material of various components. The stress must be lower than the material elastic limits with a certain factor of safety. Displacement results are used for calculating stiffness of components at specific points through dividing the applied force by corresponding displacement.

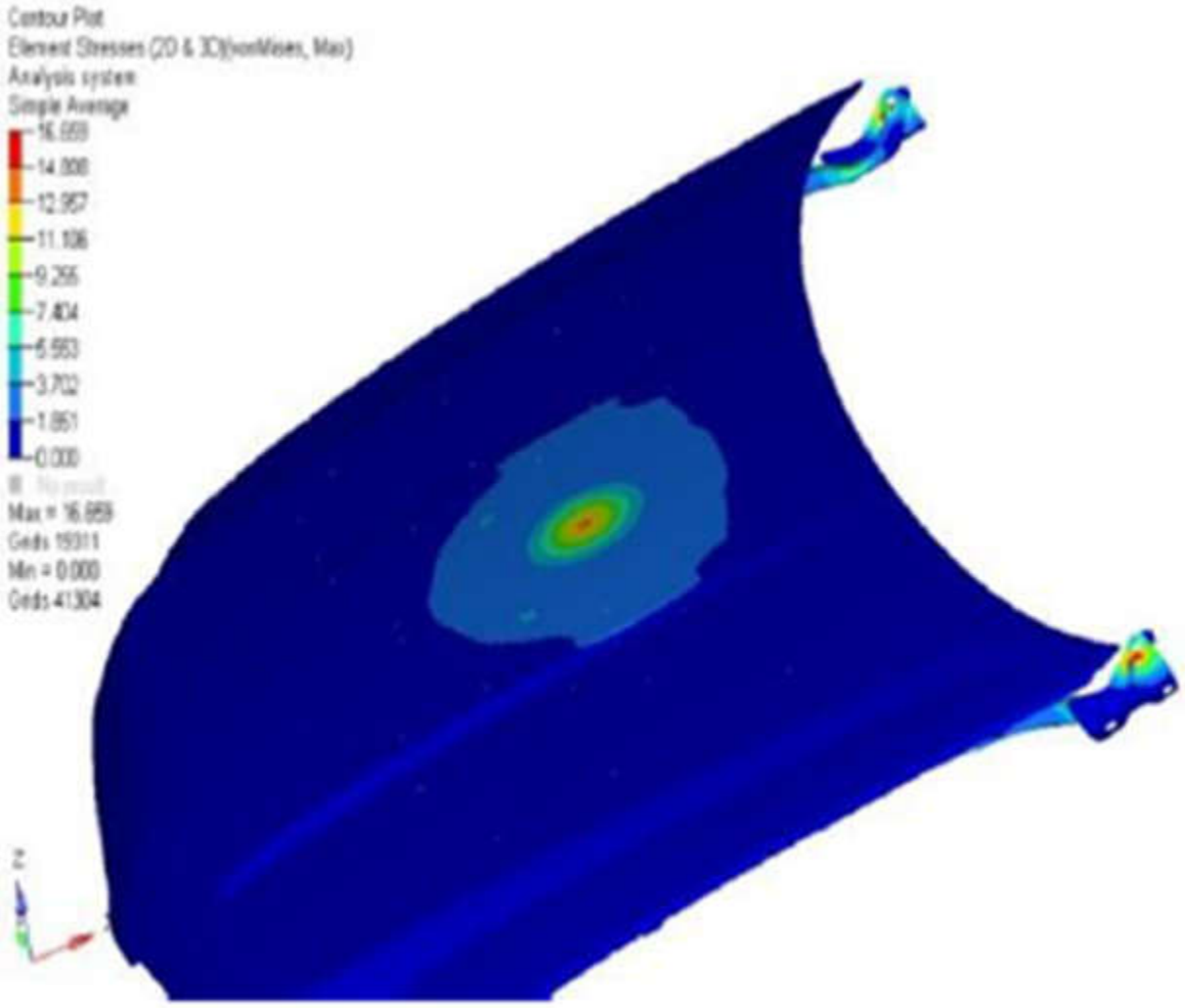
3.2. Material Selection with the Different Forces

The materials that had been chosen in this SolidWork simulation are steel (AISI 1045), aluminium alloy (Al 6061) and carbon fibre reinforced plastic (CFRP). Each material has their own properties which can be used to predict the strength, stress, displacement and deformation of the material. In this study of impact test, to evaluate the parameters of the design is by using the different forces. Maximum force that used in this simulation is 2370N. The next three forces that were used are 1870 N, 1370 N and 870 N. The force or load that has been added is directly perpendicular on the top of the car hood.

3.3. Stress Analysis Result

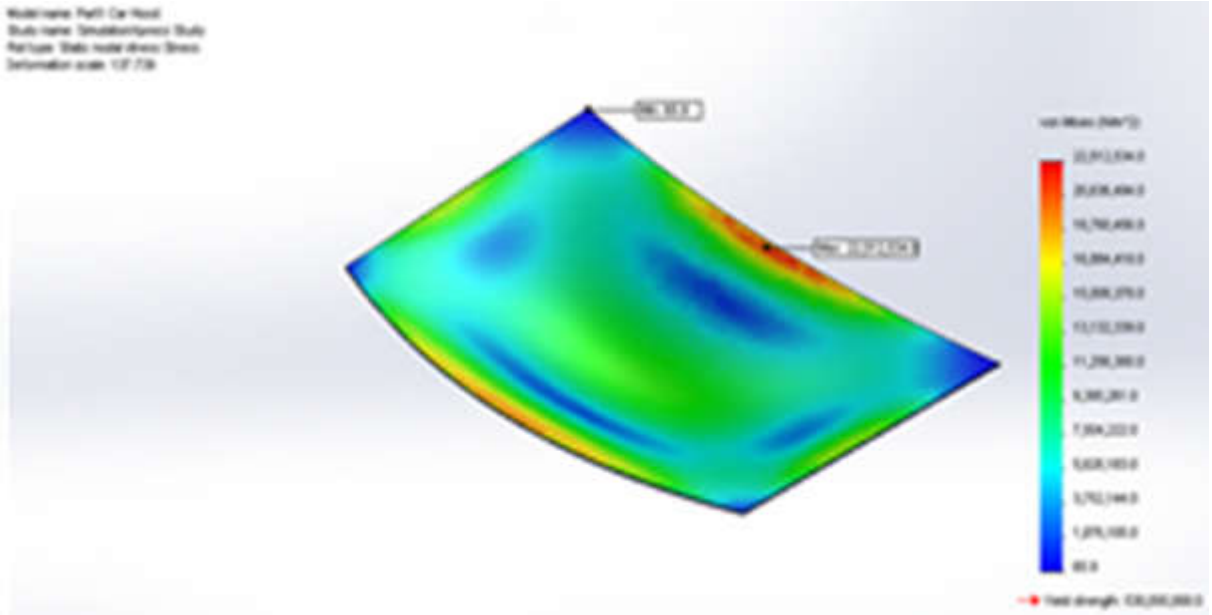


(a)

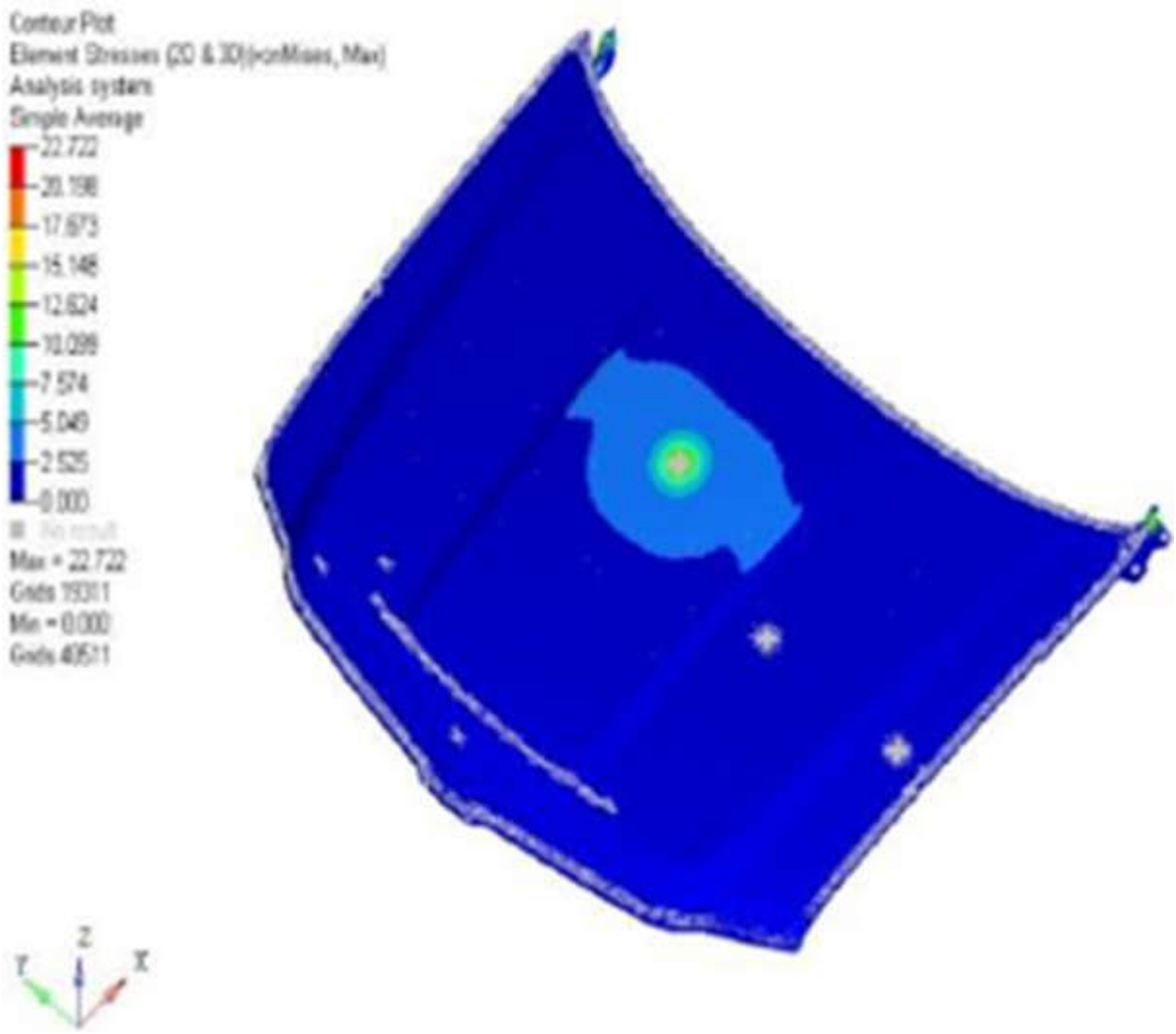


(b)

Fig.2. a) Result stress analysis of Al 6061-T6 (SS) at 2370 N and b) Al 6061 at 2370 N [31]



(a)



(b)

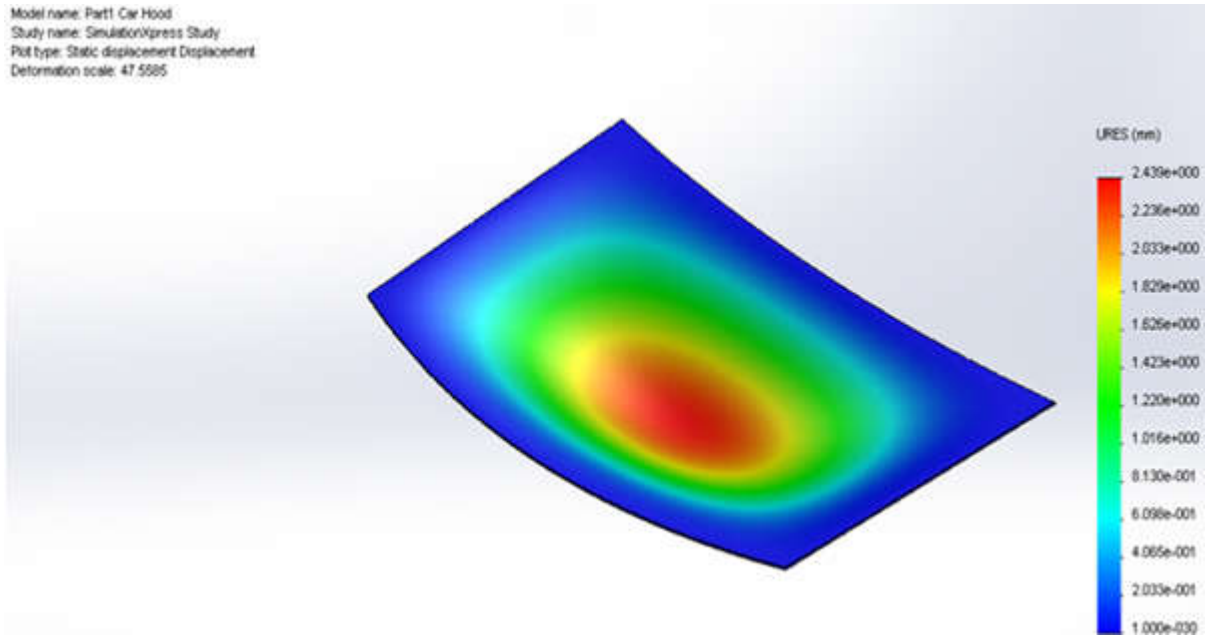
Fig.3. a) Result stress analysis of Steel AISI 1045 at 2370 N and b) Steel AISI at 2370 N [31]

Fig. 2 shows the stress result for Al 6061-T6 (SS) by using finite element analysis at 2370 N. The value of maximum stress that produced from the analysis was 2.19288e+007 N/m^2. When convert the unit of Newton/meter square to Megapascal, it will became 21.9288 MPa. Therefore, Fig. 3 shows the stress result for model of Al 6061 at 2370 N. The value of maximum stress was 16.659 MPa. The maximum stress value of finite element analysis is higher than the previous study.

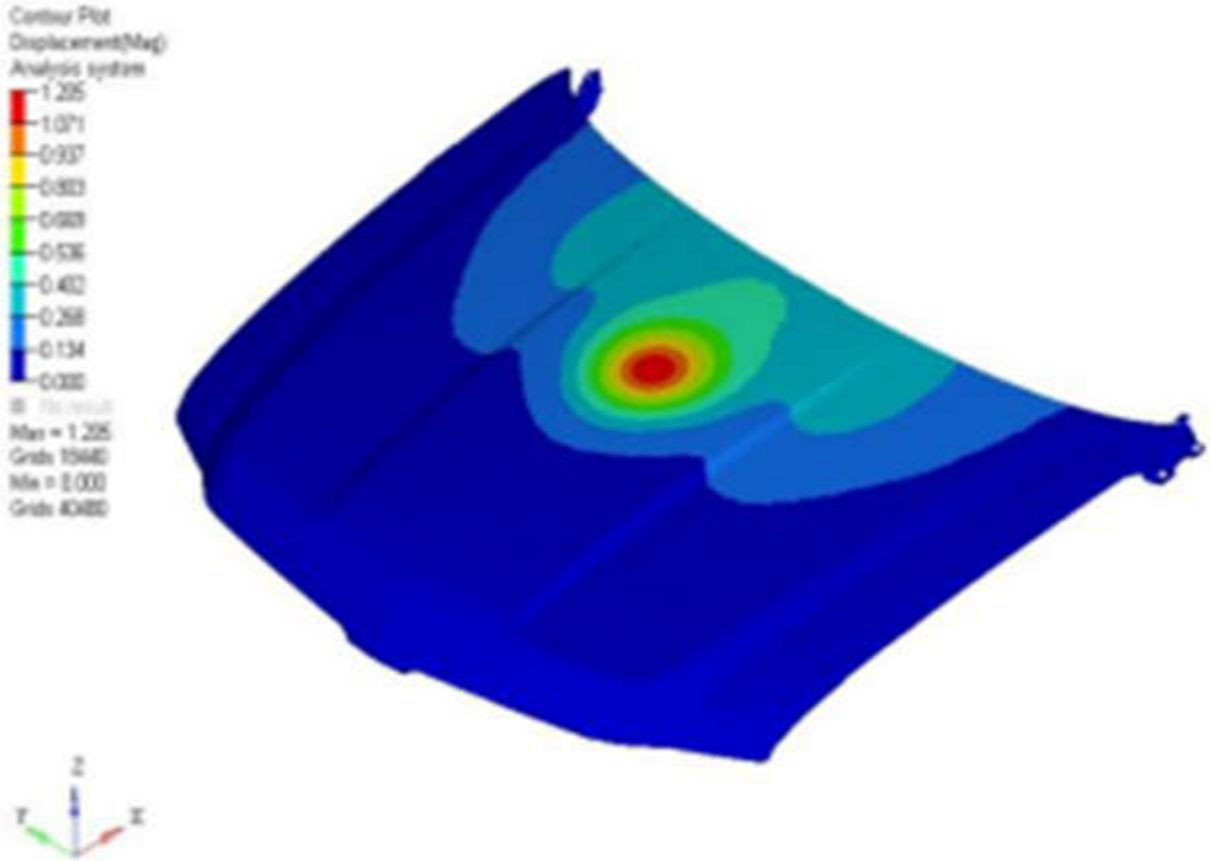
In Fig. 2 shows the result of stress of Steel AISI 1045 at 2370 N using finite element analysis. Maximum stress that produced was 22.5125 MPa. Then, Fig. 3 shows stress result for previous study of Steel AISI 1045 at 2370 N. The maximum stress that stated was 22.722

MPa. The values of both maximum stresses were almost the same.

3.4. Displacement Analysis Result

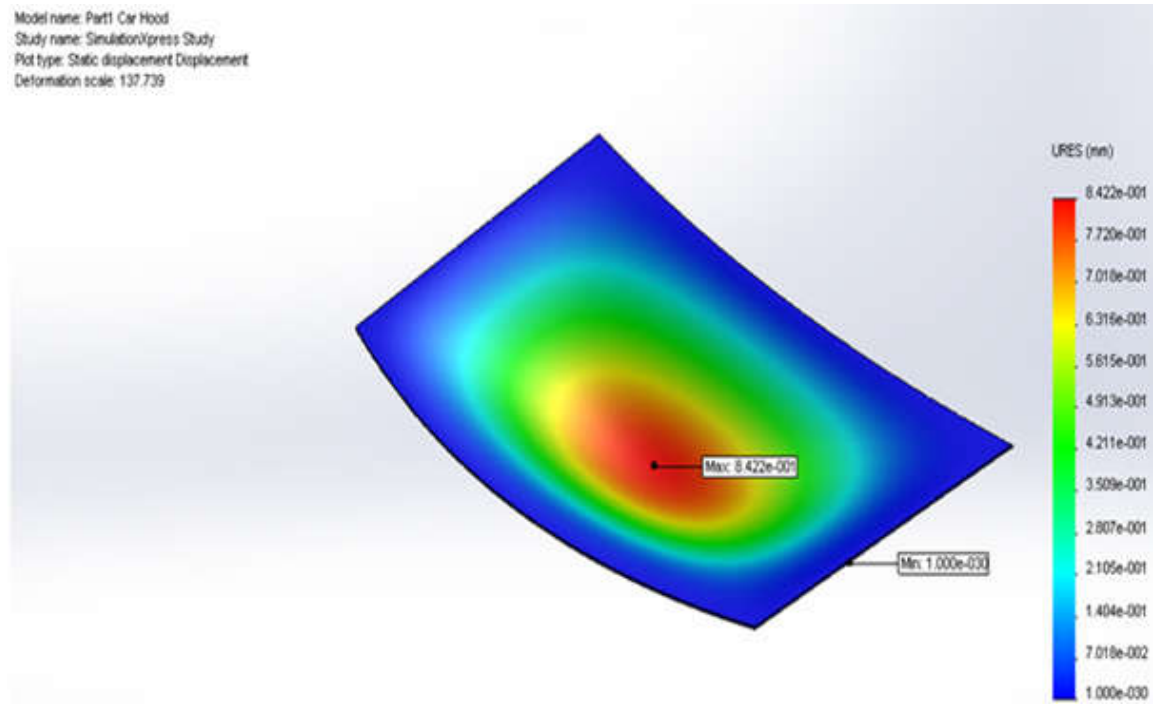


(a)

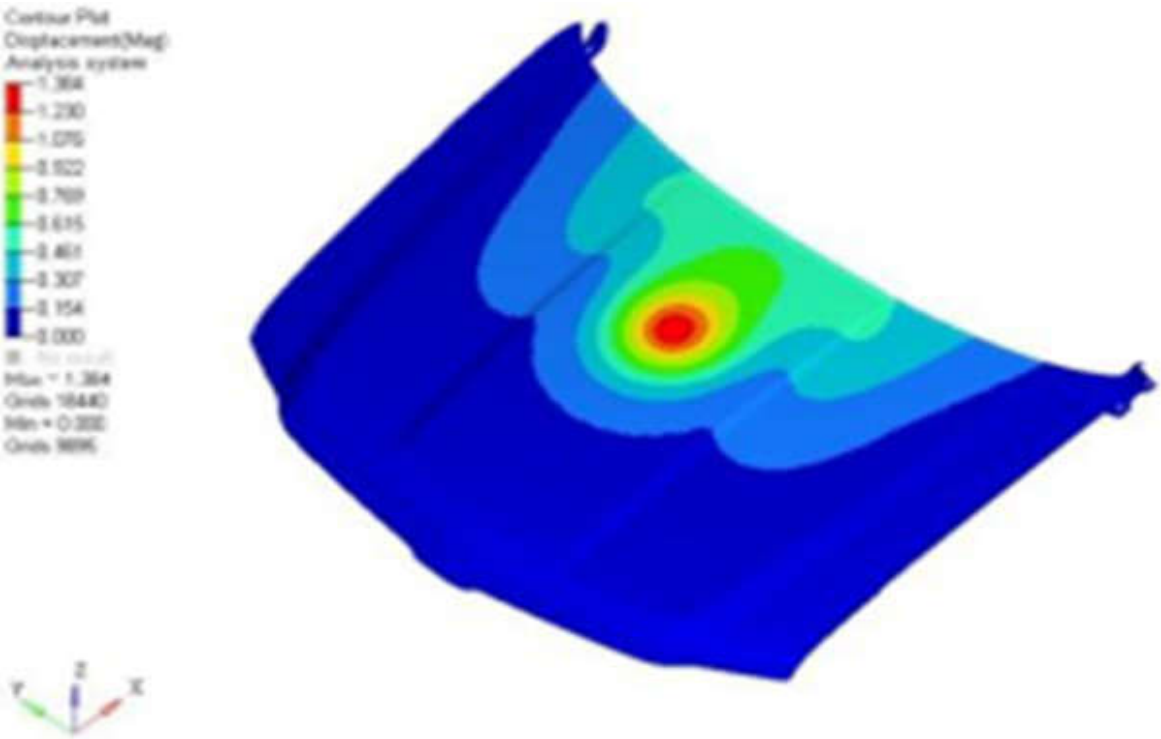


(b)

Fig.4. a) Displacement result of Al 6061-T6 (SS) at 2370 N and b) Steel Al 6061 at 2370 N [31]



(a)



(b)

Fig.5. a) Displacement result of Steel AISI 1045 at 2370 N and b) Steel AISI at 2370 N [31]

Based on Fig. 4, it shows the displacement result of Al 6061-T6 (SS) at 2370 N by using finite element analysis. The value of the maximum displacement was 2.4391 mm. While for previous study, in Fig. 4 also shows the displacement result of Al 6061 at 2370 N. Its maximum displacement was 1.205 mm. The displacement in Fig. 4 is higher than in Fig. 5. Then, at Fig. 5 shows the displacement of Steel AISI 6061-T6 (SS) at 2370 N. The value of maximum displacement that had been obtained was 0.842175 mm. Besides, in Fig. 5 shows the displacement of Steel AISI 6061 at 2370 N. The maximum displacement from the previous study was 1.384 mm.

3.5. Effect of Forces on the Displacement of Three Types of Materials

Three types of materials which are aluminium alloy Al 6061-T6 (SS), steel AISI 1045 and CFRP-Sandwich composite were set to impact test in SolidWorks simulation. During the impact test, the materials were added with four different forces to test their displacement. In Fig. 6 shows the comparison of maximum displacement between the materials due to the forces that had been added.

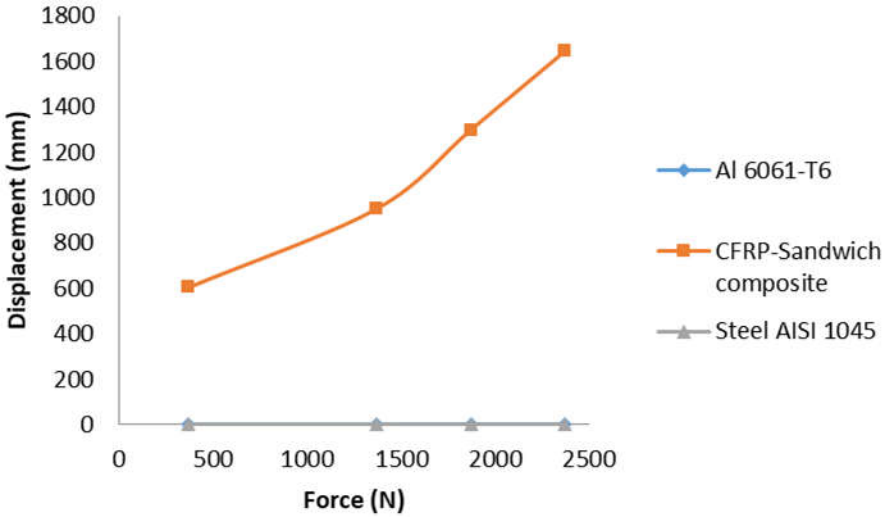


Fig.6. Graph of maximum displacement against force for three different materials

Fig. 6 illustrates the comparison between three different materials that were used in fabrication of car hood in one graph considering displacement and forces. It is shown that the maximum displacement of CFRP-Sandwich composite is more than steel and aluminium. The study proved that displacement of head form for composite hood is higher than steel and aluminium, but the amount of Head Injury Criterion (HIC) is much less than steel and

aluminium. From the graph show that maximum displacement for the Al 6061-T6 (SS) was the second highest after the CFRP-Sandwich composite. Then, the lowest maximum displacement in the graph was Steel AISI 1045. Based on the result finite element analysis force-displacement, all the curves of the graph were increasing linearly. The graph can be concluded that the more forces added to the car hood, the higher the displacement occurs.

3.6. Effect of Forces on the Stress of Three Different Materials

The comparisons are made between Al 6061-T6 (SS), AISI 1045 and CFRP-Sandwich composite with forces and stress. Four times of impact test on the car hood had been conducted. 870 N of force was added during the first impact test, then for the second impact test was 1370 N and the next two impact test were 1870 N and 2370 N. Then, the results of maximum stress were produced after running the simulation by using finite element analysis of SolidWorks.

As we all know, stress is the pressure or tension exerted on a material object. Strength and fracture toughness are two important mechanical properties. Yield strength is the measure of the stress that material can withstand before deforming. Tensile strength is a measure of the maximum stress that a metal can support before starting to fracture. Fig. 7 shows that comparison of maximum stress of the different materials based on the forces that had been added.

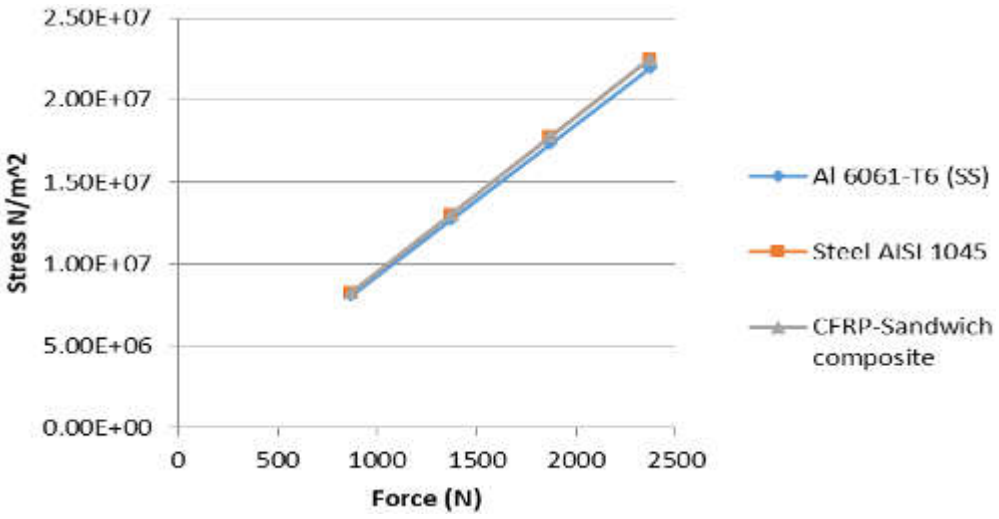


Fig.7. Graph of maximum stress against force for three different materials

Fig. 7 shows the maximum stress of each material. In general, it clearly indicates that Steel

AISI 1045 and CFRP-Sandwich composite were getting the highest value of maximum stress compared to Al 606-T6 (SS). All the reading of maximum stress between Steel AISI 1045 and CFRP-Sandwich composite were same due to the forces that had been applied except for 1870 N. Steel AISI 1045 got 1.77631e+007 N/m^2, while CFRP-Sandwich composite got 1.7763e+007 N/m^2. The lowest reading of maximum stress in the graph was Al 6061-T6 (SS). The differences value of each graph may occur due to the properties of each material. Therefore, the pattern of each graph of material was linear graph.

4. CONCLUSION

From the static analysis, the stress and displacement analysis was investigated numerically by using finite element analysis. It also was capable to simulate the impact test for car hood by using SolidWorks simulation software. The three different materials of car hood which are Al 6061 T6 (SS), Steel AISI 1045 and CFRP-Sandwich composite were examined by four different forces that impacting the car hood. 870 N, 1370 N, 1870 N and 2370 N forces had been selected for the impact test. Moreover, the simulation result by using different material in term of maximum stress and maximum displacement was successfully compared. CFRP-Sandwich composite has higher value of maximum displacement compare to Steel AISI 1045 and Al 6061-T6 (SS). From this result can predict that CFRP-Sandwich composite has high flexibility in material properties. Therefore, from the result, CFRP-Sandwich and Steel AISI 1045 almost have same value of stress. It was show that both materials have high yield strength performance compared to Al 6061-T6 (SS).

5. REFERENCES

[1] Mahasufee M. Design and analysis front hood fairing for cars. Bachelor thesis, Universiti Malaysia Pahang, 2009

[2] Höckert J A, Uyanik G. Development of a future hood concept facilitating daily maintenance and increased perceived quality. Master thesis, Gothenburg: Chalmers University of Technology, 2012

[3] Hu J. W. Advanced materials and structural engineering: Proceedings of the International

Conference on Advanced Materials and Engineering Structural Technology (ICAMEST 2015), April 25-26, 2015, Qingdao, China. Florida: CRC Press, 2016

[4] Ahmed A, Wei L. Introducing CFRP as an alternative material for engine hood to achieve better pedestrian safety using finite element modeling. Thin-Walled Structures, 2016, 99:97-108

[5] Kong C, Lee H, Park H. Design and manufacturing of automobile hood using natural composite structure. Composites Part B: Engineering, 2016, 91:18-26

[6] Gülke T, Rumpe B, Jansen M, Axmann J. High-Level requirements management and complexity costs in automotive development projects: A problem statement. In International Working Conference on Requirements Engineering: Foundation for Software Quality, 2012, pp. 94-100

[7] Bhagat R Y, More A P. A review: Analysis and optimization of car bonnet. International Journal of Innovative Research and Development, 2014, 3(1):52-55

[8] Fateme R. Development of short carbon fibre reinforced polypropylene composites for car bonnet application. Master thesis, Selangor: Universiti Putra Malaysia, 2006

[9] Elmarakbi A, Azoti W. Novel composite materials for automotive applications: Concepts and challenges for energy-efficient and safe vehicles. In 10th International Conference on Composite Science and Technology, 2010, pp. 1-10

[10] Wilson A. Vehicle weight is the key driver for automotive composites. Reinforced Plastics, 2017, 61(2):100-102

[11] Lad A, Rao A S. Design and drawing automation using solid works application programming interface. International Journal of Emerging Engineering Research and Technology, 2014, 2(7):157-167

[12] Mazlan M, Mustafa Al Bakri A M, Wahab R, Mohd Sukhairi M R, Amini M H. Comparison between thermal interface materials made of nano carbon tube (NCT) with gad pad 2500 in term of junction temperature by using CFD software, Fluent™. Materials Science Forum, 2014, 803:243-249

[13] Mohamed M, Al Bakri A M, Wahab R, Zulhisyam A K, Iqbal A M, Amini M H, Mohammad A A. Simulation of nano carbon tube (NCT) in thermal interface material for

electronic packaging application by using CFD software. Materials Science Forum, 2015, 803:337-342

[14] Mohamed M, Hashim F R, Amini M H, Janvekar A A, Razab M K, Yusuf N A, Rizman Z I. Finite element analysis of car hood for impact test by using Solidworks software in automotive application. Journal of Fundamental and Applied Sciences, 2018, 10(1S):936-955

[15] Mazlan M, Rahim A, Al Bakri Abdullah M M, Razak W, Zubair A F, Najib Y M, Bakir Azman A. Thermal management of electronic components by using computational fluid dynamic (CFD) software, FLUENTTM in several material applications (epoxy, composite material & nanosilver). Advanced Materials Research, 2013, 795:141-147

[16] Mazlan M, Rahim A, Al Bakri Abdullah M M, Iqbal M A, Razak W, Salim M S. A new invention of thermal pad using sol-gel nanosilver doped silica film in plastic leaded chip carrier (PLCC) application by using computational fluid dynamic software, CFD analysis. Advanced Materials Research, 2013, 795:158-163

[17] Gouasmi M, Ouali M, Fernini B, Meghatria M H. Kinematic modelling and simulation of a 2-R robot using solidworks and verification by MATLAB/Simulink. International Journal of Advanced Robotic Systems, 2012, 9(6):1-13

[18] Pentenrieder B. Finite element solutions of heat conduction problems in complicated 3d geometries using the multigrid method. Degree thesis, Technical University of Munich, 2005

[19] Krishnakumar S. Fiber metal laminates-The synthesis of metals and composites. Material and Manufacturing Process, 1994, 9(2):295-354

[20] Yang Q, Zhang Z, Liu M, Hu J. Numerical simulation of fluid flow inside the valve. Procedia Engineering, 2011, 23:543-550

[21] Wilhelm M. Materials used in automobile manufacture-current state and perspectives. Le Journal de Physique IV, 1993, 3(C7):31-40

[22] Ponnusamy B. Applications of finite element analysis in mechanical system design. 2016, https://www.researchgate.net/profile/Barath_Ponnusamy/publication/282357590_APPLICATIONS_OF_FINITE_ELEMENT_ANALYSIS_IN_MECHANICAL_SYSTEM_DESIGN/links/560e477408ae967420111df4.pdf

[23] Mohammed S D, Desai H. Basic concepts of finite element analysis and its applications

in dentistry: An overview. Journal of Oral Hygiene and Health, 2014, 2(5):1-5

[24] Meng J, Liu Y, Liu R. Finite element analysis of 4-cylinder diesel crankshaft. International Journal of Image, Graphics and Signal Processing, 2011, 3(5):22-29

[25] Al-Momani E, Rawabdeh I. An application of finite element method and design of experiments in the optimization of sheet metal blanking process. Jordan Journal of Mechanical and Industrial Engineering, 2008, 2(1):53-63

[26] da Silva E P, da Silva F M, Magalhães R R. Application of finite elements method for structural analysis in a coffee harvester. Engineering, 2014, 6(03):138-147

[27] Purohit R, Khitoliya P, Koli DK. Design and finite element analysis of an automotive clutch assembly. Procedia Materials Science, 2014, 6:490-502

[28] Abdullah O I, Schlattmann J. Finite element analysis of temperature field in automotive dry friction clutch. Tribology in Industry, 2012, 34(4):206-216

[29] Maeno T, Hasegawa J. Development of a finite element model of the total human model for safety (THUMS) and application to car-pedestrian impacts. In International Technical Conference on Enhanced Safety of Vehicles, 2001, pp. 1-10

[30] Logan D. L. A first course in the finite element method. Boston: Cengage Learning, 2011

[31] Bhaskar N, Rayudu P. Design and analysis of a car bonnet. International Journal of Current Engineering and Technology, 2015, 5(5):3105-3109

How to cite this article:
Mohamed M, Din MFM, Hashim FR, Amini MHM, Masri MN, Bakar MBA, Razab MKAA, Rizman ZI.Comparison between carbon fibre reinforced plastic, aluminium alloy and alloy steel for car hood application for impact test by using finite element analysis. J. Fundam. Appl. Sci., 2018, *10(2S), 463-485.*