DESIGN OPTIMISATION OF AUTOMOTIVE COMPONENT THROUGH NUMERICAL INVESTIGATION FOR ADDITIVE MANUFACTURING

Mohammad Al Bukhari Marzuki
Mechanical Engineering Department
Sultan Azlan Shah Polytechnic, 35950 Behrang, Malaysia
Email: mohammad@psas.edu.my

Mohammad Firdaus Mohammed Azmi
Mechanical Engineering Department
Sultan Azlan Shah Polytechnic, 35950 Behrang, Malaysia
Email: firdaus_azmi@psas.edu.my

Rafidah Laili Jaswadi
Mechanical Engineering Department
Sultan Azlan Shah Polytechnic, 35950 Behrang, Malaysia
Email: rafidah@psas.edu.my

ABSTRACT

Additive manufacturing technology plays a major role in the Industry 4.0 where the manufacturing method supports the digitisation of development, testing and fabrication of product before reaching the consumer. The use of lightweight design and material have become inevitable in the modern world. The needs for improved fuel economy created importance in using lightweight and high strength design and material for automotive component structure. Producing lightweight and high strength parts and component also can be utilised in hybrid or electric vehicle where the lighter component will be translated into a longer driving range. Design optimisation method in finite element analysis can be utilised to ensure that the component or part for automotive application is designed according to the specification. In a combination of the additive manufacturing method and design optimisation, automotive parts manufacturers are capable of produced lightweight component design and at the same time performed similarly or better compared to original design. To archive a specific target for designing a lightweight component, a computational method is utilised, commercially available design tool Inventor and Shape Generator respectively. The optimisation procedure is based on the parametric investigation against the critical structure performance in reference to the maximum stress and deformation value from the applied load on the component. Optimised part design typically are unsuitable for conventional manufacturing which usually produced the bio-inspired or organic shape, therefore additive manufacturing technique is employed to produce such parts or component. The benefits of utilising additive manufacturing technique will translate into faster parts production to the end user and reduction of material used in producing the component. The initial cost of acquiring a suitable 3-D printer is one of the disadvantages in utilising additive manufacturing technique in automotive parts and component production. The outcome of the investigation can be utilised to assess the aptness of the optimisation method and additive manufacturing technique in producing automotive components or parts.

Key words: Design optimization, finite element analysis, additive manufacturing

INTRODUCTION

Automotive parts and component manufacturers nowadays faced a number of challenges in producing their product sustainably especially when the automotive industry moving towards producing energy efficient vehicle and the limitation/costly natural resource to acquire. One of the ways to sustainably use raw material in producing automotive components is to introduce design optimisation during as early as the product development stage. Combined with additive manufacturing technology, the outcome of the component not only sustainably use raw material but also with improved structural performance. Topology optimisation is a method to generate optimal topology in relation to the material distribution on the initial design. Finite element analysis is utilised to define the design space and variable which allowing the design variation based on the parameters such as weight, deformation and stress level on the design.

Finite element analysis (FEA) is one of a design tool to simulate the performance of design statically and dynamically. A study demonstrated that FEA is utilised to develop, analyse and optimized design based on the result generated from the simulation (Marzuki et al. 2015 & Marzuki et al. 2015). An analysis conducted by (Venkata, 2013) shows that both CATIA V5R16 and ANSYS 11.0 are utilised to perform stress analysis and design optimisation involving engine piston. The outcome of static structural analysis and design optimisation concluded that the piston design could be optimised especially in reducing weight around the lower part of the piston which is essential for improving efficiency and optimising raw material consumption. A material optimisation analysis on steering component is investigated by (Kumar, 2017) utilising CAD geometry and further studied using ANSYS version 16.0. The finite element analysis software package ANSYS is utilised to analyse the deformation, stress and elastic strain of the steering component design with three types of material being compared to for comparative design

analysis. The significant amount of weight reduction when the component is applied with S-glass epoxy material with the stress and deformation is within the safe limit of the material. Apart from applying the design optimisation process at the component level, the optimisation by utilising FEA can be applied to larger system automotive system such as body, frame and chassis. As studied by (Rajput, 2016), an ATV chassis structure is analysed by using AutoDesk Inventor software package for its elastic stress and strain behaviour under loading condition. The loading condition is applied on the strut attachment point to simulate the weight of the ATV during operation. The outcome of the FEA analysis is utilised to make design changes to ensure good weight to power ration and safety of the driver of the ATV.
A suspension system is utilised to reduce the amount of vibration from uneven road surface to the cabin which ensuring the comfort of the driver and passenger. A suspension system is considered among the critical system in vehicle design and construction. This statement is supported by some research and investigation regarding the suspension system to ensure the safety and comfort of the passenger. As investigated by (Kumar, 2016), a sub-component of a suspension system is investigated and optimised using finite element analysis method. The intention of analysis is to reduce material usage and strengthening for sheet metal thickness for the lower control arm. From the analysis, the material thickness is reduced as much as 0.6 mm from the original design without compromising the material stress limit.

Moreover, the structural rigidity of the optimised design is better in comparison with the initial design (Kumar, 2016). Finite element analysis (FEA) software package can be employed to perform topology optimisation based on the condition set by the user. As investigated by (Muhammad, 2012), FEA software HyperWorks are utilised to re-design automotive parts to reduce the overall weight of the component. The pre-processing setup is completed using HyperMesh while the solving process is performed using RADIOSS. The optimisation process reduced 24% mass from the original design which is significant in terms of designing and producing a product through sustainable practices.

A study conducted by (Sharma, 2014) utilising CREO 2.0 and ANSYS Workbench for geometry creation and FEA analysis respectively. ANSYS's topology optimisation is employed to investigate the best use of material for the design. From the optimisation process, the mass of the knuckle is reduced by 19.35% with both maximum stress and local deformation is within the safety limit of the design. Topology optimisation for crane hook design is another example of utilisation of a combination of CAD and FEA software (Thejomurthy, 2018). The geometrical modelling is created from CATIA V5 software while the FEA analysis and design optimisation are employed using HyperWorks software package. By using Opti-struct solver in the HyperWorks, the optimised geometry based on weight, stress and deformation condition are established at the 30th iteration which provides 35% weight reduction in comparison with the initial model (Thejomurthy, 2018). Knuckle is a crucial part to ensure safe handling of a vehicle. The design of knuckle utilised in a vehicle is to function similar to a hinge which allows movement of connecting upper and lower arm in the suspension system. A knuckle design study by (Yadav, 2017) utilised computational design and analysis method in determining the structural integrity of the knuckle design. As mentioned by (Yadav, 2017), the knuckle design should have the strength to overcome numerous loads during operation. The knuckle is designed and analysed further using FEA workbench in CATIA software package. The design of the knuckle is based on steel material which presumably comparable to material construction of knuckle in real knuckle application in a vehicle suspension system. Form the analysis, the maximum stress developed at the knuckle joint area with a peak value of 201 MPa which well below the maximum allowable material stress value of 400MPa (Yadav, 2017). The initial design and FE analysis utilising CATIA can be used as a benchmark for further FEA and experimental analysis.

Design optimization is a process to create the designs that fully exploit the capability of a newer manufacturing process which involves an additive manufacturing process. It also can explore the design space to be more efficiency. Additive manufacturing enables to produce the complex functional parts geometry, multi-material and individualized mass production. It has an ability to produce optimized geometries with near perfect strength to the weight ratios. A study by (Gore, 2017) explored a different load condition on an automotive component which the component experienced when in operation. Among the loading condition applied to the steering knuckle are bumping, braking, steering and combined loading. Each of every loading condition produced different stress plot and the stress analysis is utilised as a basis of optimisation.

The integration of topology optimisation through finite element method and additive manufacturing are discussed in academia and industry especially the integration method is an important aspect in Industry 4.0. A study by (Mirzendehdel, 2016) provides an in-depth analysis of utilising the integration of design optimisation and additive manufacturing through support structure design. The outcome from the study shows that the effectiveness of using finite element analysis supporting topological optimisation for additive manufacturing. Although, further research is needed to explore additive manufacturing constraint in producing a product. Design optimisation is widely used for aerospace industries which are largely related to weight reduction on aircraft parts and components. The optimisation method also can be applied to wing design to reduce ground sonic boom as analysed by (Alonso, 2007). Automotive manufacturer nowadays is in need to improve product performance to meet challenging raw material shortage, strict emission control and stringent government law. Designing and analysing automotive components are getting economical in parallel with the affordability of computer technology. As a result, almost 100% automotive component is developed through computational methods.

Although there is numerous literature concerning the research on utilising FEA and optimisation for automotive component development, most of the study fails to address the design optimisation for rocker arm design in a vehicle suspension system. In this article, a rocker arm design is selected for FE analysis to assess the structural deformation and stress pattern and the design will undergo design optimisation iteration to determine the utmost design parameter in reference to the earlier structural analysis result.

FINITE ELEMENT METHOD

The Finite Element Method (FEM) is a numerical technique for solving the problem of engineering and physics. It is also referred to as Finite Element Analysis (FEA). FEA is a numerical method for finding approximate solutions to boundary value problems. Generally, there are three steps in FEA, starting with pre-processing, solving and post-processing. The first step in pre-processing is to create a 3D model and define the material properties. The mesh will be generated uniformly for the entire structure then the boundary conditions are applied such as supports or load. In solving process, depending on the type of analysis, a solver is selected to find a solution for the condition applied to the model. In this study, the solver is automatically selected by the AutoDesk Inventor software package. The final step is post-processing which was the result are produced based on analysis type defined during the pre-processing procedure. As a basis of this study, stress plot and deformation plot are selected to determine the location of stress and deformation when the load is acting on the rocker arm.

MATERIAL PROPERTIES

Table 1: Material Properties

Young's Modulus	68.9 GPa
Tensile Yield Strength	276 MPa
Ultimate Tensile Strength	310 MPa
Poisson's Ratio	0.33
Shear Modulus	26 GPa
Density	2.7 g/cc

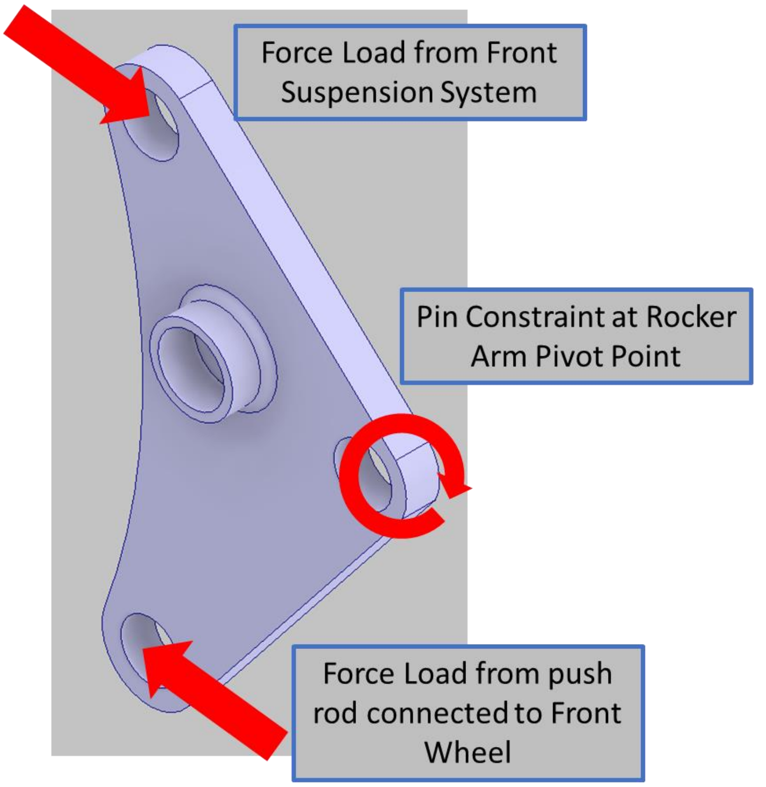
MESH GENERATION

The CAD model of the rocker arm is converted into STEP format file and imported into AutoDesk Inventor software for analysis. The rocker arm model is applied with Tetrahedral 10 type of element with ten nodes positioned and four physical points at each element. The element is selected due to its ability to produce high accuracy solution utilising the second order function. The element size is fixed at 0.015 mm throughout the model. This element size setup created 112240 elements and 166067 nodes for the whole structure. The number of elements and nodes are acceptable considering high-quality mesh created by the selection of element type and size.

LOAD AND BOUNDARY CONDITION

In this study, the load conditions are applied at both push rod and absorber mounting with the value of force applied are equivalent to the condition where the vehicle is cruising on a flat road surface. Pin constraint support is applied on the pivot point of the rocker arm. The rocker arm is allowed to move in rotary motion with the limitation from both load condition at two mounting points. The rocker arm is modelled partially to utilise the symmetry boundary condition which would considerably save the computational resource during the solving process. The symmetry boundary condition would effectively free up element limitation and the extra available element are applied to the area of interest related to maximum stress and deformation value. Figure 1 shows the schematics of the boundary and loading condition of the rocker arm model.

Figure 1: Model schematics for load and boundary condition



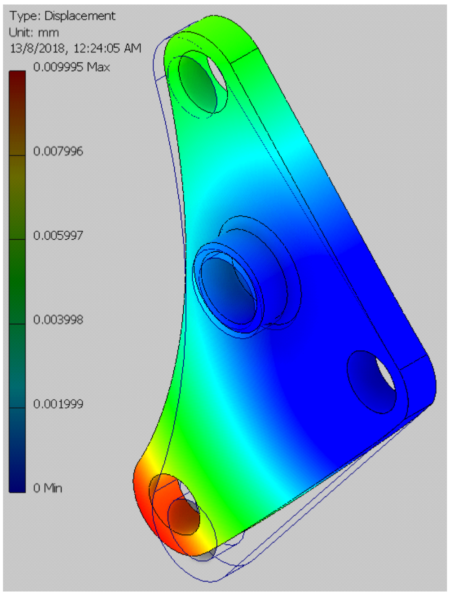
TOPOLOGY OPTIMIZATION

In topology optimization, the loads and constraints are defined first then a volume that the part will occupy is defined. When the boundary condition and loading are applied in FEA solution, the analysis started with the full design space. Then the material is progressively removed by using mass target reduction until the final iteration is stopped. The final design is assumed to be optimized to a defined level of efficiency at that mass target. From the stimulation, the software generates a part shape that satisfies the loads and constraints while minimizing the design goals of minimum part weight and maximum part stiffness.

ANALYSIS OF ROCKER ARM

The initial analysis of the rocker arm by applying the loading and boundary condition found that the local stress concentration at the pivot point with a maximum value of 5.425 MPa. The local stress concentration located at the pivot point is mainly caused two force acting on both sides of the mount points, with the upper mount point connected to the spring-absorber assembly and the lower point connected to the push rod. The loading condition produced a large deformation at both mounting points of the rocker arm with the maximum deformation recorded at the lower mount point which is linked to push-rod and the lower arm assembly of the front wheel. The maximum displacement at the lower mount point recorded a 0.0099 mm of deformation in comparison with an unloaded model. The deformation for the initial rocker arm design is illustrated in Figure 2.

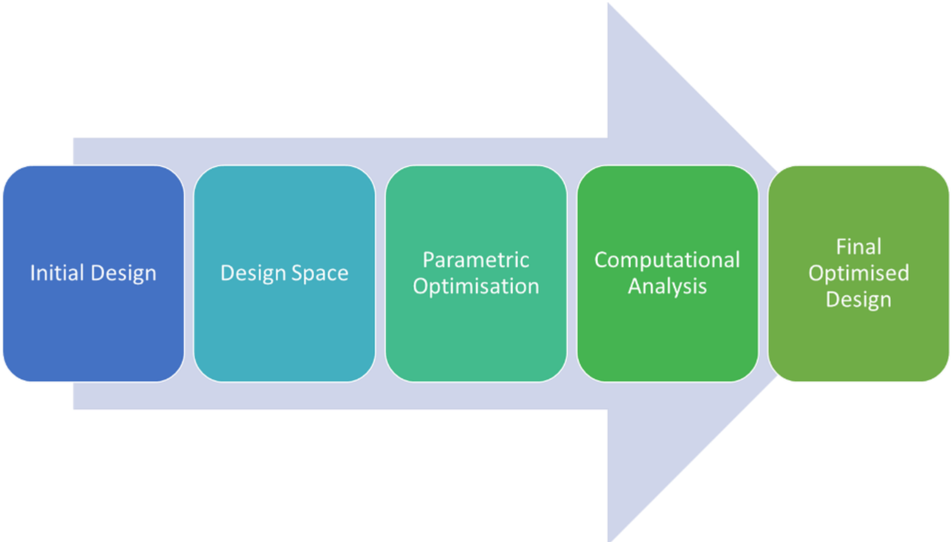
Figure 2: Deformation plot for initial rocker arm design.



DESIGN OPTIMISATION OF THE ROCKER ARM

The optimisation method for analysing the best weight and stress value of the rocker arm design is determined by the required design space, selection of design critical point/area and design optimisation criteria. The process flow of the rocker arm design optimisation is illustrated in Figure 3. The entire process of design optimisation is conducted using the computational method in AutoDesk Inventor's Shape Generator module.

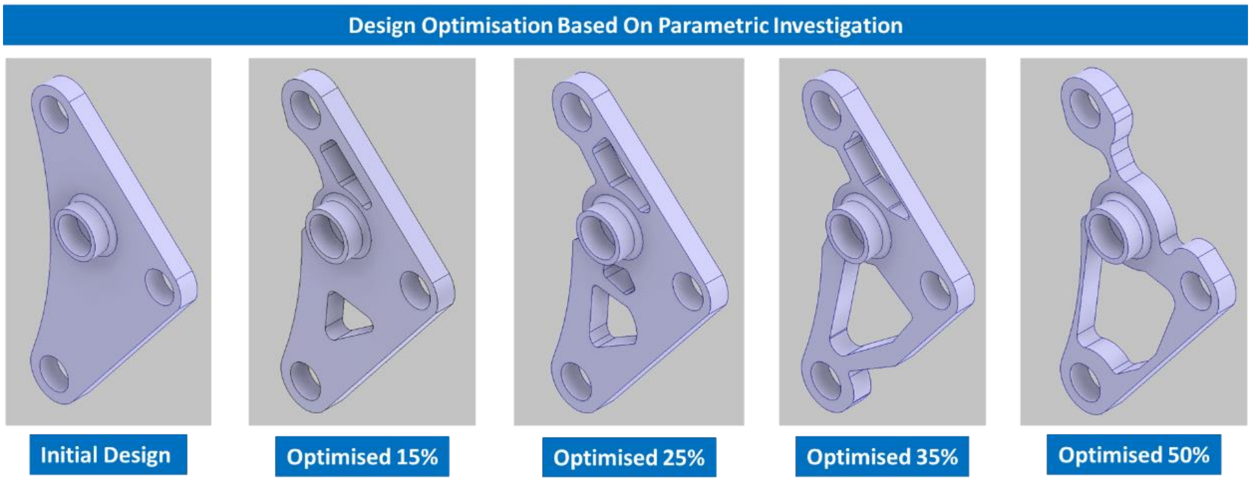
Figure 3: Process flow for rocker arm design optimisation



Design space is the criteria for product design to function in the required standard or limitation. This is determined during the initial design of the component concurrent with other parts design which makes a larger assembly design. Once the design space is determined, the critical area in the design is selected and exempted from the optimisation process. This step is to ensure the rocker arm design will able to produce functional design after the optimisation solution is completed. Four critical areas are selected in the rock arm design, centre support column, pivot point and both lower and upper mounting points. The objective for design optimisation is selected according to the criteria for the rocker arm design.

The criteria for rocker arm design is weight optimisation, therefore four weight reduction for the design 15%, 25%, 35% and 50% are selected for the optimisation process. Subsequently, the four design criteria are analysed using similar boundary and loading condition as per initial design. The deformation and maximum stress plot are compared to determine the best design in terms of weight saving and the amount of stress/deformation level. The weight criteria for design optimisation are shown in Figure 4.

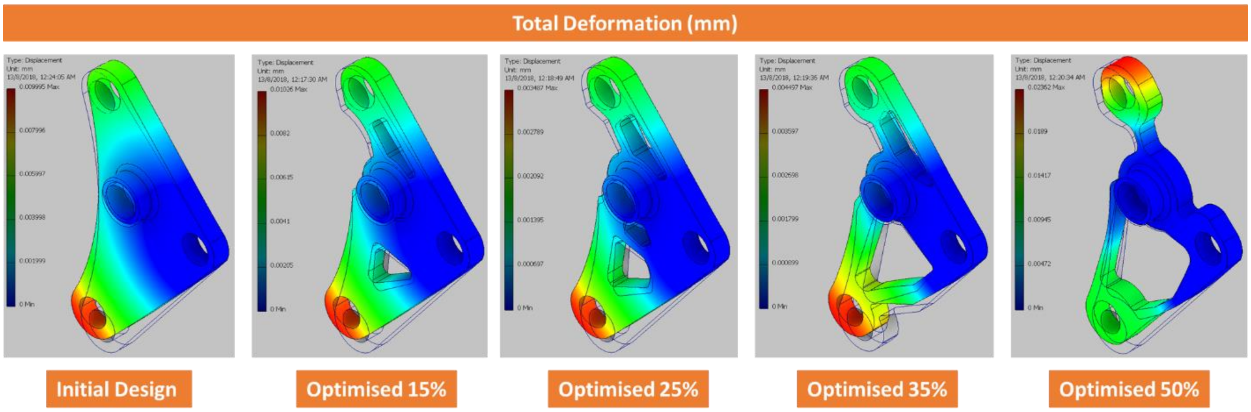
Figure 4: Rocker arm optimisation based on weight criteria.



DESIGN OPTIMISATION RESULT

Design optimisation requires in-depth analysis especially in determining the area of critical to the component. This will ensure the design will perform similarly to the initial design if not better. The result from AutoDesk Inventor's Shape Generator as shown in Figure 4 is analysed structurally similar to the boundary and loading condition to the initial analysis on the original rocker arm. The result from the analysis is plotted with deformation and stress plot and the plotted result is compared with the initial design. The deformation and stress plot for the optimised design are shown in Figure 5 and Figure 6 respectively.

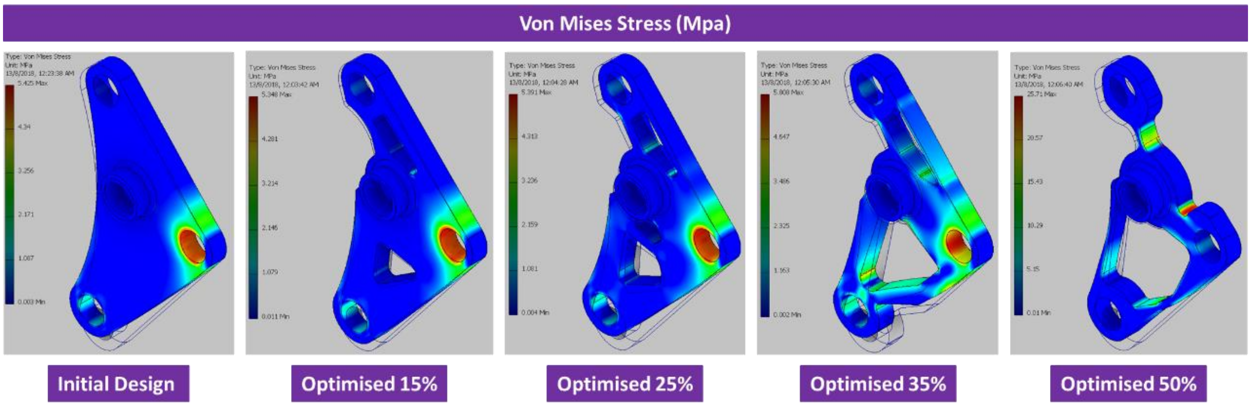
Figure 5: Deformation plot for different weight criteria.



The total deformation plot shown in Figure 5 started with the plot from the initial design on the left-hand side, followed by 15%, 25%, 35% and 50% weight reduction. The maximum deformation location on the initial design, 15%, 25% and 35% located at an almost similar position on the lower mount point where the push rod is connected. The 50% model recorded the maximum deformation at the upper mount point which is used to link the spring-absorber assembly. This is likely due to the 50% model loose most of the material surrounding the upper mount and caused the maximum deformation shifted to the upper mount point.

The deformation value is increasing in parallel with weight reduction where the least amount of deformation is recorded on the initial model whereas the highest displacement readings are on the 50% model. This is caused by the removal of material on the optimised models which lead to losing most of its load-bearing capability.

Figure 6: Stress plot for different weight criteria.



Finite element analysis (FEA) is a numerical method that use elements and nodes to represent the geometry of the rocker arm model. Although the result from FEA is an approximation from a real experimental analysis, the resulting plot can be used as a guide in providing insight towards the real physical analysis. Figure 6 shows the von Mises Stress Plot for rocker arm model with the initial stress model are on the left-hand side followed by 15%, 25%, 35% and 50% weight reduction model.

The initial, 15% and 25% produced similar stress concentration, where the stress peaked at around the pivot mount point. As the weight is reduced at 35%, new local stress concentration developed especially around the lower mount point for push rod mechanism in addition to the pivot mount point. When the weight of the rocker arm model is further reduced to 50%, much of the stress concentration developed at the pivot point at earlier model are relocated to the upper neck area between the central support column and the pivot mount points. The relocation of stress concentration is largely caused by the removal of load bearing segment on the upper section between upper mount points and the pivot mount point which provide much of the support from the applied loads.
Stress value for the 1st three models are almost similar with lowest are at the 15% model. The 50% model recorded a sharp increase in stress value with an increase as much as 5 times higher in comparison with the initial model. A stated earlier, the value of stress is closely associated with the load-bearing capability of the 50% mode which loses most of the material from the optimization process.

CONCLUSION

A total of 4 design are developed by utilising parametric optimisation with weight reduction ranging from 10% to 50% from the initial design. The initial design recorded maximum stress of 5.425 MPa with the maximum stress located at the pin constraint boundary condition. The maximum deformation for initial design is located at the push rod mounting with a recorded maximum value of 0.00995 mm. The optimised design with 35% of weight reduction offered the best result in terms of maximum stress value, maximum deformation and weight reduction in comparison with other models.

The result from this investigation needs further validation by using other optimisation method and verified from experimental analysis. The optimised design is suitable to be manufactured using a conventional and additive manufacturing method, but further model clean-up is needed for conventional method due to the limitation of the machining process. Apart from that, the optimised model also offers reduced raw material consumption which directly improved the sustainably of the product development process.

REFERENCES

Alonso, J. J., LeGresley, P. & Pereyra, V. (2009). Aircraft Design Optimization. Mathematics and Computers in Simulation. 1948 – 1958.

Anupam Raj Jha, Rakesh Jaiswal, Anush Karki, Ankit Basnet, Pawan Jaiswal, Saurav Rajgadia, Debayan Das, Rabindra Nath Barma. (2016) Design and Finite Element Analysis of Knuckle Joint Using CATIA and ANSYS Workbench. International Journal of Research in Mechanical Engineering. Vol. 4, Issue 3, 1-5.

Gore, S. R., Gund, K. K., Patane, P. M., Mohite, N. V., & Chimote, C. V. (2017). Topology optimization of Automotive Steering Knuckle using Finite Element Analysis. International Journal of Current Engineering and Technology. 122 – 125.

Kumar, C. S., Kumar, B.K., Rao, J. K., Kumar. R. A. (2017). Design Optimisation of a Steering Knuckle Component Using Conservative Method of Finite Element Analysis. International Journal for Research in Applied Science & Engineering Technology. Vol. 5, Issue 11, 525 – 532.

Kumar, S. A., Balaji, V., Balachandar, K., Kumar, D. P. (2016). Analysis and Optimization of Lower Control Arm in Front Suspension System. International Journal of Chemical Sciences. Vol. 14, Issue 2, 1092 – 1098.

Marzuki, M. A. B., Bakar, M. A. A. & Azmi, M. F. M. (2015). Designing Space Frame Race Car Chassis Structure Using Natural Frequencies Data from Ansys Mode Shape Analysis. International Journal of Information Systems and Engineering. Vol. 1 No. 1, 01 – 10.

Marzuki, M. A. B., Halim, M. H. A. & Mohamed, A. R. N. (2015). Designing Space Frame Race Car Chassis Structure Using Natural Frequencies Data from Ansys Mode Shape Analysis. American Journal of Engineering and Applied Sciences. Vol. 8, Issue 4, 538 – 548.

Mirzendehdel, A. M. & Suresh, K. (2016). Support Structure Constrained Topology Optimization for Additive Manufacturing. Journal of Computer-Aided Design. 1 – 37.

Muhamad, W. M. W., Sujatmika, E., Idris, M. R., Ahmad, S. A. S. (2012). An Optimization Analysis on an Automotive Component with Fatigue Constraint Using HyperWorks Software for Environmental Sustainability. International Journal of Mechanical and Mechatronics Engineering. Vol. 6, Issue 8, 1395 – 1399.

Rajput, N. & Mehta, G. (2016). Modelling & Finite Element Analysis (FEA) of ATV Chassis to Enhance Efficiency. International Journal of Advances in Engineering & Scientific Research. Vol. 13, Issue 2, 01 – 17.

Sharma, M. P., Mavewala, D. S., Joshi, H., Patel, A. D. (2014). Static Analysis of Steering Knuckle and its Shape Optimization. IOSR Journal of Mechanical and Civil Engineering (IOSR - JMCE). 34 -38.

Thejomurthy, M. C. & Ramakrishn, D. S. (2018). Topology Optimization and Analysis of Crane Hook Mode. Journal of Engineering Research and Application. 60 – 64.

Venkata, C. R., Murthy, P. V. K., Krishna, M. V. S., Rao, G. M. P. (2013). Design Analysis and Optimization of Piston using CATIA and ANSYS. International Journal of Innovative Research in Engineering & Science. Vol. 1, Issue 2, 41 – 51.

Yadav, S., Benade, S., Angchekar, S., Dhokle, V., Kolhapure, R. (2017) Design And Analysis of Knuckle Joint by Using FEA. International Journal of Current Engineering and Scientific Research. Vol. 4, Issue 6, 70 – 77.