Australian Journal of Basic and Applied Sciences, 9(30) Special 2015, Pages: 22-35

ISSN:1991-8178

Australian Journal of Basic and Applied Sciences

Journal home page: www.ajbasweb.com

Design Optimization of Automotive Fog-Lamp Bracket for Weight and Cost Reduction

S. Norazlan, A.A. Faieza and Z. Norzima

Department of Mechanical and Manufacturing Engineering, Faculty of Engineering, Universiti Putra Malaysia, 43400.Serdang, Selangor, Malaysia.

ARTICLE INFO

Article history:
Received 25 July 2015
Accepted 30 August 2015
Available online 9 September 2015

Keywords:
Design Optimization, FEA, Parametric Optimization, Weight & Cost Reduction, DOE.

ABSTRACT

There are growing concerns over energy security and the impacts of climate change. With these concerns in mind; one of the options that car-makers have nowadays is to focus on weight reduction as one of the method to achieve fuel efficiency vehicle. The objective of the study is to evaluate the performance of existing fog-lamp bracket and to come out with an optimized bracket; lighter in weight and cheaper on the piece part cost. The geometrical model of the fog-lamp bracket initial design is re-modeled by using Solidworks (CAD software) and FEM or Finite Element Model generated in Solidworks Simulation is used for required dynamic and static analyses. The lowest 10 natural frequencies, distribution of stresses and displacements under static loading of the fog-lamp bracket, as well as the stationary behaviors under harmonic excitations are examined. In the next step, these results are taken into account for optimization problem formulation. The optimal parameters of the fog-lamp bracket are found using parametric optimization in the Solidworks Design Study. The results showed that the stress and natural frequency of the fog-lamp bracket changed little in comparison with the original design, an amount which was within the allowable engineering parameters, while the material cost was reduced by 70% and the part weight was reduced by almost 80% due to material changed.

To Cite This Article: S. Norazlan, A.A. Faieza and Z. Norzima., Design Optimization of Automotive Fog-Lamp Bracket for Weight and Cost Reduction. ***Aust. J. Basic & Appl. Sci.,*** *9(30): 22-35, 2015*

INTRODUCTION

There are growing concerns over energy security and the impacts of climate change. One important and effective policy option is to raise the minimum standards for light-duty passenger vehicle fuel economy. U.S Congress has passed the Energy Independence and Security Act of 2007 (EISA), which increased the target fleet-wide average fuel economy standard to 35 miles/gallon in 2020 (Cheah, 2010 and Shiau *et al*., 2009). Same goes to Europe OEMs, they are also under intense pressure to cut fleet CO2 emissions and the goal for 2020 is 95 grams/kilometer (Transport and Environment, 2013).

Consumers worldwide have also watched the cost of fuel continue to fluctuate throughout the year. Seeing this as an issue, consumers are now looking for fuel efficient car as an option. Sakaguchi has performed a survey as part of his study and stated that 76% of the respondents answer "yes" to consider a purchase of fuel efficient car, the answer unveiled the considerable potential needs for fuel efficient cars in near future (Sakaguchi, 2010). Cheah *et al.* outlined three options to achieve fuel efficiency and one of it is to "reduce vehicle weight and size" (Cheah *et al*., 2007).

Car manufacturer are also having difficulty on the increase of material cost. There is a need to reduce the amount of material used in constructing their vehicle components or to replace costly material such as steel to a cheaper material without jeopardizing the existing performance. This is the challenge that car manufacturer need to observe to maintain relevant in the demanding market nowadays. With these three combine issues; emissions and environmental impact, unstable fuel price and higher material cost, the research need is fully justified in serving the environment, the industry and the consumers on achieving better quality and cost effective products.

To demonstrate the effective use Finite Element Analysis (FEA) and Design Optimization, an automotive fog-lamp bracket is chosen as a medium for this study. The reason of selecting fog-lamp bracket in this study is because the existing fog-lamp bracket is made of steel was having corrosion issue due to exposure to weather condition. The study is looking for material substitution for steel which is more agile to weather condition. Fog-lamp which is

Corresponding Author: S. Norazlan, Department of Mechanical and Manufacturing Engineering, Faculty of Engineering, Universiti Putra Malaysia, 43400.Serdang, Selangor, Malaysia.

shown in Figure 1 is mainly attached to the vehicle bumper using a bracket made of sheet metal or plastics.

Fig. 1: Fog-lamp assembled on car front bumper (source: Automotive Lighting).

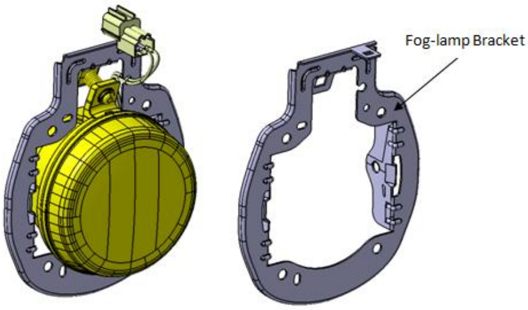


Fig. 2: Fog-lamp and Bracket (Source: Automotive Lighting).

The mounting bracket (Figure 2) which is used to mount the light module to the bumper should be a rigid structure where the force at the required frequency can be transferred from the vibrating structure efficiently. Mounting brackets are usually made of highly damped, stiffened and light weight materials such as plastic, steel sheet and aluminum.

2. Literature Review:

A. Parametric Optimization:

Albert *et al*. (2007) in their research paper discuss the implementation of Finite Element Models (FEM) in robust design process. They claimed that variation of components parameters must be included in simulations and optimizations. They emphasized on the utilization of CAD models to analyze the influence of variations using an automated process from the geometric model to the evaluation of the analyses results which they believed modern CAD software and optimization tools nowadays me*et al*l requirements for such automation.

B. Case Studies on Parametric Optimization:

Ooi *et al*. (2013) perform analysis on a new propose portal axle unit which is installed on a higher ground clearance vehicle and off-road vehicle. In this paper, a hollow shaft with a rib at both ends was proposed and the torsional stress of the model was determined using finite element analysis (FEA). Results from the FEA analysis were validated by experimental.

There were five parameters considered in the torsional strength analysis of the rib which; the hollow shaft thickness, rib thickness, spokes depth, rib fillet radius and the number of spokes. TAGUCHI method is used to determine the optimum set of parameters of the hollow shaft with a rib; 4 mm hollow shaft thickness, 4 mm rib thickness, 10 mm depth of spokes, 2 mm rib fillet radius and 4 spokes. The study found that the hollow shaft thickness affects the torsional strength of the hollow shaft with a rib compared to the other four parameters. The optimized shaft has an improvement in strength of 13.77% but an increase of 20% in weight compared to the benchmark shaft testing.

Covill *et al*. (2014) in their study had outline a FE model using beam elements to represent a standard road bicycle frame. The model simulates two (2) standard loading conditions to understand the vertical compliance and lateral stiffness characteristics of 82 existing bicycle frames from the bicycle geometry project and compare these characteristics to an optimized solution in these conditions. Perhaps unsurprisingly smaller frames (490 mm seat tube) behave the most favorably in terms of both vertical compliance and lateral stiffness, whereas the shorter top tube length (525 mm) and larger head tube angle (74.5°) results in a laterally stiffer frame which corresponds with findings from literature. The optimized values show a considerable improvement over the best of the existing frames, with 13% increase in vertical displacement and 15% decrease in lateral displacement when compared to the best of the analyzed frames. The model has been developed to allow for further develop to include more detailed tube geometry, additional analysis of more frame geometries, materials options and analysis of other structural characteristics.

C. Studies & Achievement on Cost & Weight Reduction:

Mahanty *et al*. (2001) in their study "Analysis and Weight Reduction of a Tractor's Front Axle" had analyzed the new design of the front axle of tractor for thirteen (13) different Certification Test load conditions. There is no field failure reported on the current design and the study performed by mean of comparison purpose of the proposed new design only. Based on the FEA results, the front axle design was revisited for weight optimization and ease of manufacturability and five proposed designs were evaluated for selected worst load cases based on the current design. Analysis results of new models yielded displacements and stresses close to the current design. The increases in stresses were close to 15 %, while for the displacement; no significant results are achieved but met the existing structural requirement. On the other hand, there was a significant reduction in weight (approximately 40 %) and the proposed models did not involve a lot of welding process, thus significant savings of manufacturing cost was observed. All the components used in the assembly founded to be more cost effective like smaller diameters bearing, smaller knuckle size and etc. The reduction in manufacturing cost and weight drastically reduced the cost of the new design of Front Axle. This analysis work showcases the implementation of finite element analysis (FEA) and design optimization as a technique for cost reduction, in terms of materials and manufacturing.

3. Research Methodology:

The methodology shown (Figure 3) in this research is an engineering design framework based on design tools and computer aided design and engineering (CAD/CAE). This framework also integrates not only design of performance and cost issues but also manufacturability aspect. Based on these three aspects and with the use of analytical tools it is possible to envision and develop advance product designs that comply with market and usability objectives, while achieving high performance product and lower total costs.

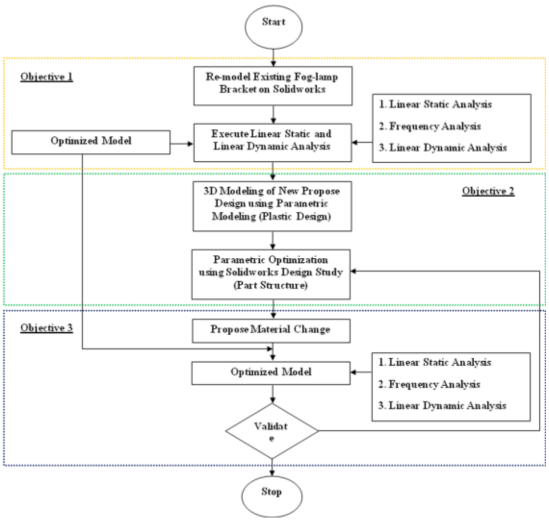


Fig. 3: Research Methodology.

The optimization of the bracket will be carried out using Parametric Optimization approach. Design or 3D modeling will be done using Solidworks while pre-processing meshing and post-processing analysis (Linear Static & Frequency and Dynamic-Harmonic Analyses) will be carried out using Solidworks Simulation. Solidworks Parametric "Design Study" which is based on DOE will be used to define design variables and the result will be published in iterations where the best design option is then been selected.

4. Performance Evaluation:

The process of evaluating the existing metal bracket started with re-modeling the 3D model from 2D drawing provided by Automotive Lighting Malaysia. 3D modeling was performed using Solidworks to make used the parametric design capability offered by the software for later simulation activity. Figure 3 and 4 show the 2D drawing and 3D modeling of the existing metal bracket.

Figure 5 and 6 illustrate the general size of the bracket of the bracket and the assembly view of bracket to the lighting module.

The bracket is manufactured by sheet metal stamping. JIS G3141-2005 SPCC (Cold Rolled Steel

Sheet) 1.2 mm thickness is used as materials. The main inertial properties of the bracket such as mass, moments of inertia, center of mass, as well as volume, surface area and others are calculated. Mass of the initial bracket is m0 = 140.8 grams and volume of the part is v0 = 17,892.02 mm3.

Bracket model is meshed with parabolic tetrahedral elements using Solidworks Simulation. The resulting mesh is shown in Figure 4.5. The FE mesh is generated with curvature based mesh (max elements size is 7.1 mm, min element size is 1.4 mm, element size growth ratio is 1.5) which ensures accurate representation of the test model.

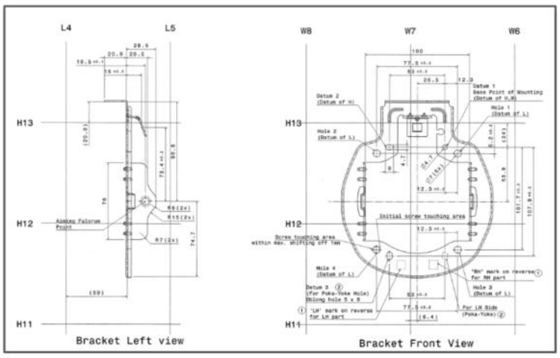


Fig. 3: 2D drawing of existing metal bracket.

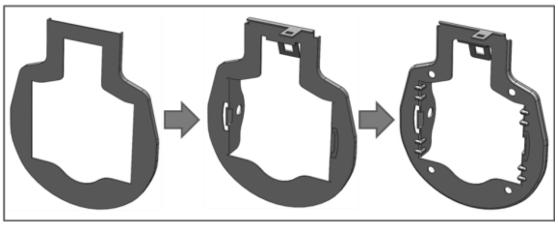


Fig. 4: 3D part generation using Solidworks.

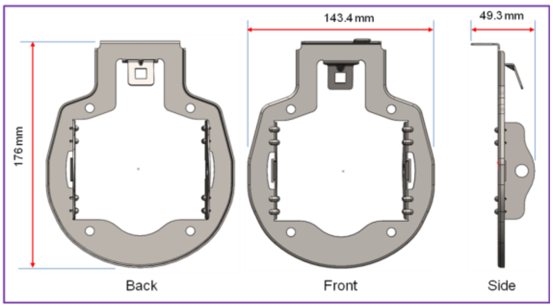


Fig. 5: Part general size and ISO view of the Fog-lamp bracket.

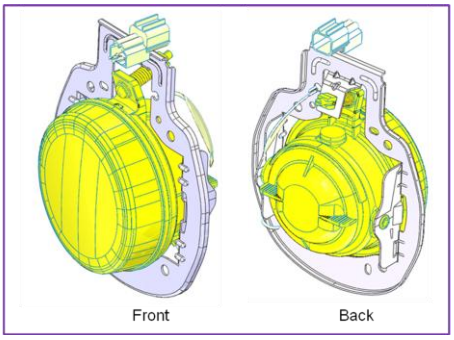


Fig. 6: Lighting module assembled to Fog-lamp bracket.

Fig. 7: Fog-lamp bracket meshed model.

Finite Element (FE) analysis is used to evaluate different responses of fog-lamp bracket: Maximum von Mises stress in the material from impact loading; maximal displacements and accelerations at the characteristic points of bracket in stationary and transient vibration processes due to harmonic excitations and natural frequencies of the fog-lamp bracket.

A. Strength Calculation:

Generally the strength of fog-lamp bracket design is checked using special vibrostands. The initial prototype of the bracket is subjected to different dynamic loads. Vibrostability and vibration strength of the bracket are checked on excitations in the frequency domain from 10 to 410 Hz. One of the main experiments is a test of shock resistance of the bracket design under acceleration level a = 10 g. Such experiments require significant material and time expenses and for optimization purposes the computer based design check must be used.

Figure 8 shows the initial model analysis setup in Solidworks Simulation. The bracket is fixed to the bumper at 4 locations (fixed mounts) and the light module attached at 3 locations which presented as a remote load and acceleration acts in the vertical direction (shown as downward arrow).

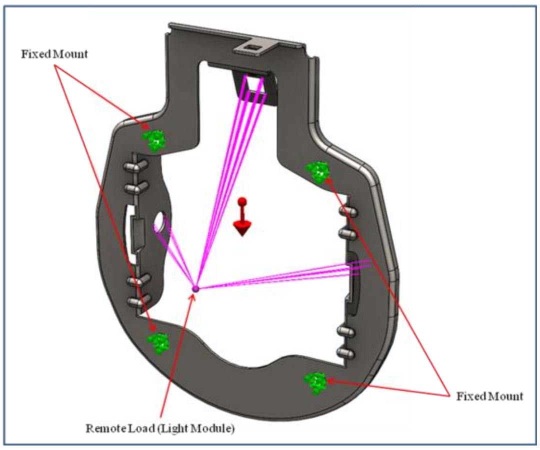


Fig. 8: Model setup for FE analysis.

For the initial design of the bracket the maximum von Mises stresses from impact loading are presented in Figure 9. The most loaded places, where von Mises stress level is greater than 7.2 MPa, are shown in red. Maximal stresses can be seen concentrated near the bottom 2 corners of the light module fixing flanges area: the stress level reaches as high as 7.9 MPa. Other areas of the bracket are significantly less stress levels.

The side view of the deformed shape (scale up 10K times) of the bracket and resultant displacements URES are presented in Figure 10. The upper part of bracket is twisted and has maximum displacements. Maximum deformations can be seen on the upper flange area as indicated in Figure 10.

B. Frequency Analysis:

Frequency analysis is carried out to find natural frequencies of the initial bracket model and evaluate possible resonance in the case of external excitation. The same FE mesh for model as considered before is used. The numerical solver direct Sparse of Solidworks Simulation is used for calculations of the 10 lower natural frequencies. The obtained results (Table 1) show that fundamental frequency of the fog-lamp bracket is sufficiently high f1 = 840.56 Hz.

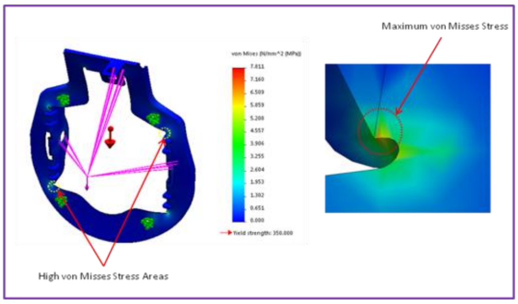


Fig. 9: Most loaded places of the bracket from impact loading.

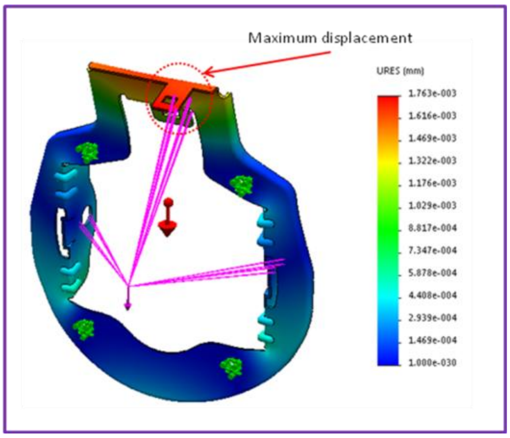


Fig. 10: Most displaced places of the bracket from impact loading.

Table 1: 10 fundamental frequencies of the existing fog-lamp bracket.

Mode No.	Frequency (Rad/sec)	Frequency (Hertz)
1	5281.4	840.56
2	6362	1012.5
3	6455.4	1027.4
4	7522.6	1197.3
5	9287.7	1478.2
6	11469	1825.4
7	13445	2139.9
8	13831	2201.3
9	14953	2379.8
10	16141	2568.9

The obtained mode shapes for the bracket are shown in Figure 11. The red color shows the regions with maximal and blue with minimal displacements. There is a high density of natural frequencies are acting on the system. Consequently, any of the lower frequencies can play an important role in the bracket vibrations.

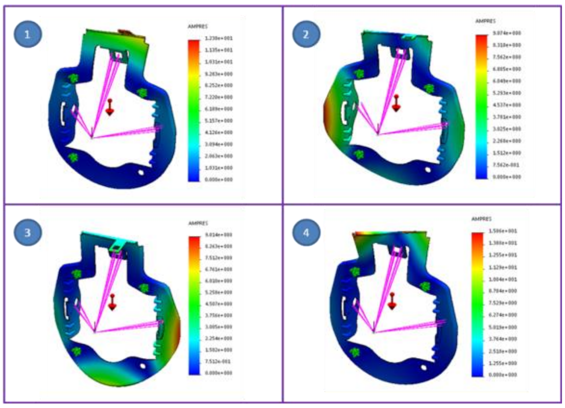


Fig. 11: The four lower natural modes of the fog-lamp bracket.

C. Harmonic Analysis:

Possible vibration levels of the fog-lamp bracket are analyzed for harmonic excitations in the frequency domain from 800 to 2600 Hz. The modal damping ratio was set to 0.002 which is general damping for sheet metal (Janssen Precision Engineering). The sensor points are clustered around each natural frequency to capture the response accurately at these frequencies neighborhood. The responses are calculated at the prescribed locations of the bracket to reduce necessary computational recourses. This point is situated at the bulb center of the light module (Figure 12: Point 1). As was shown before in the static calculation, maximum stresses occur on bottom flange corner which indicated as Point 2 (Figure 12: Point 2) and high stresses also observed on the top flange corner which indicated as Point 3 (Figure 12: Point 3).

The resultant displacements are presented in Figure 13. There were significant magnitudes of vibration obtained on the ninth resonance frequency.

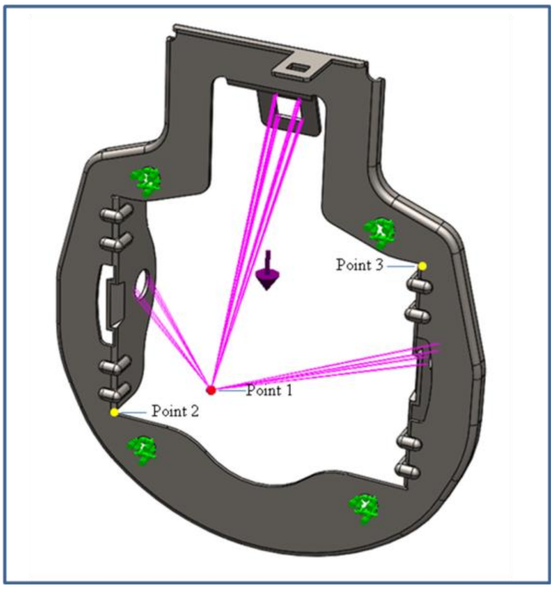


Fig. 12: Definition of the location points for calculation of responses.

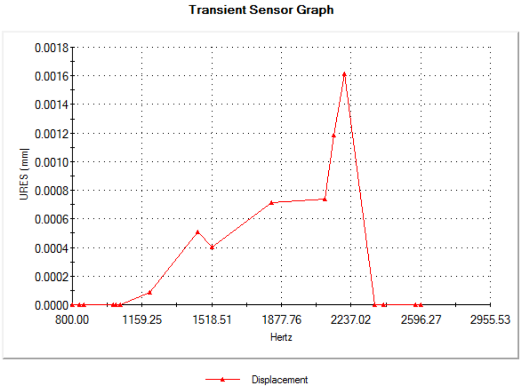


Fig. 13: Resultant displacements at the defined “Point 1” of the bracket.

Table 2: Results comparison between Linear Static and Dynamic Analysis.

Analysis Results	Linear Static Analysis	Dynamic Analysis
Resultant Max Displacement (Point 1)	1.87E-05 mm	2.00E-03 mm
Resultant Max Von Misses Stress (Model)	7.81 MPa	40.08 MPa

Comparison of maximum von Misses stress shows the highest level of stress is near to “Point 2” of the bottom light module fixing flange. It occurs between 8 and 9 natural frequencies when the stress value is 5 times bigger than in previously discussed linear-static case (Table 2).

Figure 14 - Figure 16 show the displacement components in the directions X, Y and Z

respectively. The results indicate that displacement in the direction Z are bigger on the second resonance frequency and reach 2.00E-03 mm. While displacement on X and Y directions also show the same trend, both have peak displacement on the second resonance frequency (X: 9.93E-04 mm and Y: 5.77E-05 mm).

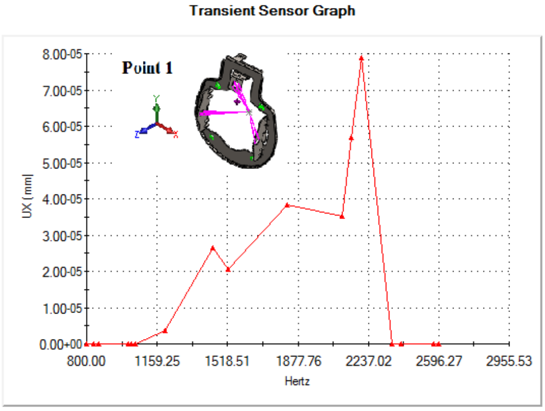


Fig. 14: UX displacement at “Point 1” of the bracket.

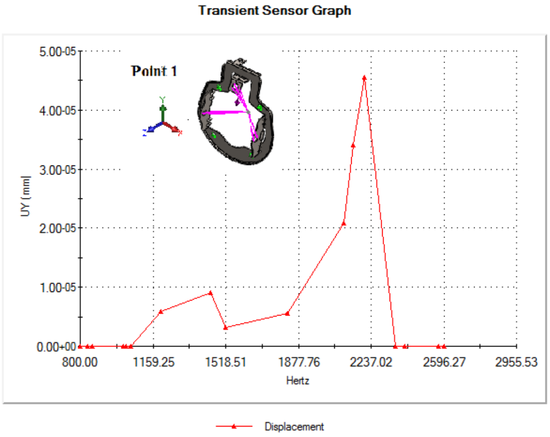


Fig. 15: UY displacement at “Point 1” of the bracket.

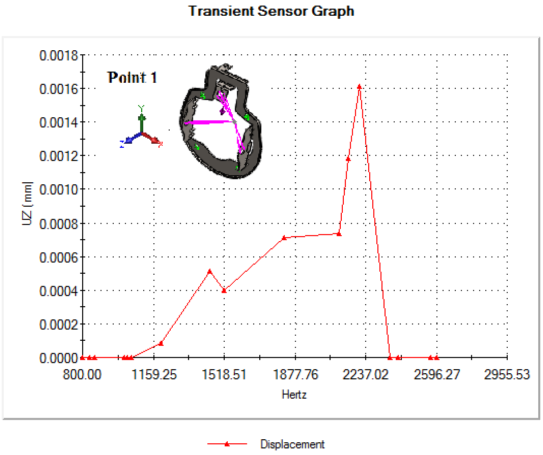


Fig. 16: UZ displacement at “Point 1” of the bracket.

D. Analysis Summary:

It’s a rule of thumb in the Automotive industry to keep the frequency of all the brackets attached to cross car beam (Steering Member) above 35Hz and the calculated stress should be less than 50% of the yield stress (Singh, 2002).

a. 50% of the yield stress for SPCC = 350/2 = 175 MPa
b. Frequency of first mode should be above 35Hz (CAE Requirement).

The maximum von Misses stress is 44.08 MPa as shown in Table 4.5 and frequency for first mode was 840.56 Hz as shown in Table 4.3. From these results, it shows that the maximal von Misses stress (44.08 MPa) was less compare to 50% of the yield stress (175 MPa) while the first mode frequency which is 840.56 Hz is almost 25 times more than the required frequency (35Hz). To conclude, the existing Fog-lamp bracket can be considered as "overdesigned" and there is a room for improvement to further decrease the mass which eventually decrease the cost of material. From this point the calculated stresses and frequency of first mode was used as baselines for the Optimization of the Fog-lamp Bracket.

Table 3: Summary of results of existing fog-lamp bracket.

Analysis Results	Linear Static Analysis	Dynamic Analysis
URES Max Displacement (Point 1) mm	1.87E-05	2.00E-03
UX Max Displacement (Point 1) mm	-1.23E-06	9.93E-04
UY Max Displacement (Point 1) mm	8.70E-07	5.77E-05
UZ Max Displacement (Point 1) mm	1.86E-05	2.93E-03
Max Von Misses Stress (System) MPa	7.929	44.08
Max Von Misses Stress (Point 2) MPa	6.854	50.38
Max Von Misses Stress (Point 3) MPa	5.238	20.23
1st Mode Frequency (Hz)	840.56	

5. Design Optimization:

Design Study uses parametric, feature-based modeling and automatic regeneration capabilities to automate the optimization process. It is based on the Design of Experiments (DOE) method; the study finds the optimum design according to goals or to objective functions, design variables, and constraints. The analysis runs iterations of the values and tests the optimum combination to meet the specific goal.

For the optimization of the fog-lamp bracket, the process is divided into 4 stages. The first stage is to define a basic optimized part features, second is for weight reduction third is for strength improvement and fourth is to test the part on several different plastic materials. All the stages will be discussed in the details in Sub-section C.

A. Parametric Modeling:

Starting point for the parametric solid model is often an approximate size and shape sketch of part being created, as dimensions might be added later on to vary the actual size and shape of the geometry. Parametric solid model is an intelligent representation of a part, it is important to analyze and detail out every part before modeling to determine the most efficient sequence for creating the features.

Parametric solid model or 3D data is surely a smart representation of a component, it is vital to analyze and plan every part before modeling to establish the most effective sequence for creating the features. Poor modeling strategies will consume time of part creation and cause difficulty during part editing. Features have to be created to permit for maximum part flexibility and variation. Rather than distinguishing the completed solid model as a large solid mass, it needs to be observed as a composition of features that are prone to be modified.

Figure 17 shows the dimensioning activity of the bracket according to parametric modeling principle. Selected dimension or features that governed the thickness and the shape of the parts were then input into Solidworks Design Study for the optimization process.

Before proceeding for the optimization process, initial material for the proposed fog-lamp bracket is set to PBT-GF30 which is a common material used in automotive industry. PBT is a semi-crystalline material and has excellent chemical resistance, mechanical strength, electrical properties (high dielectric strength and insulation resistance), and heat resistance, all of which are stable over a broad range of environmental conditions. PBT has very low moisture absorption. To further improve the strength, PBT which is used in automotive is filled with glass fiber thus increase its tensile strength from 50 MPa up to 170 MPa (depends on the percentage of the glass fiber). Below table shows the mechanical and thermal properties of PBT-GF30 used as initial material for the bracket prior to design optimization process.

Mass of the initial proposed bracket based on the initial set material is mP^0 = 44.85 grams and volume of the part is vP^0 = 28,747.27 mm3.

B. Strength and Frequency Analysis (P0):

To establish a basic understanding of the initial proposed part, the plastic bracket is subjected to the linear static analysis to determine the strength. The initial test setup is same as the setup done on this existing metal bracket these include the fixture points, load and all the response point locations.

Figure 18 and Table 5 show the static analysis of the initial proposed model (P0) and the result of the static analysis. By comparing existing metal bracket and P0 plastic bracket, it shows that the maximum resultant displacement is slightly higher. Detail of the results are show per below table 4.9.

The initial (P0) plastic bracket is also subjected to frequency analysis and the 10 modes results are show in Table 6.

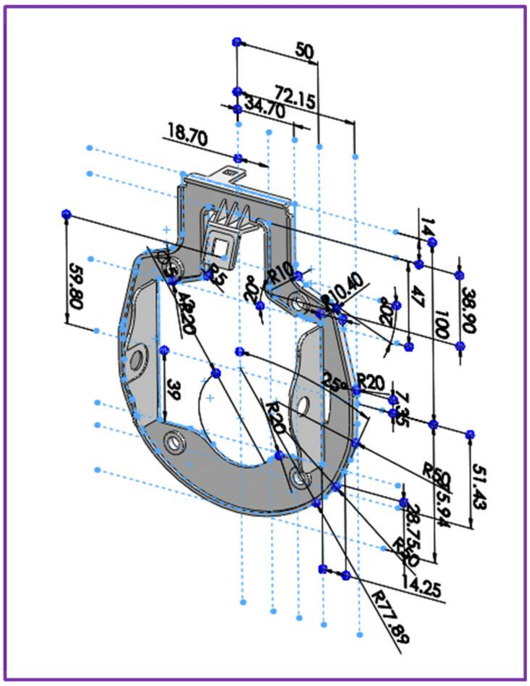
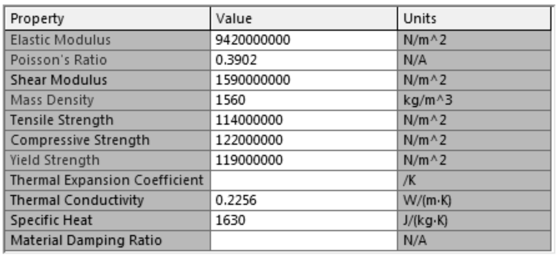


Fig. 17: Part dimensioning of parametric modeling.

Table 4: PBT-GF30 Material Properties in Solidworks.



Property	Value	Units
Elastic Modulus	9420000000	N/m^2
Poisson's Ratio	0.3902	N/A
Shear Modulus	1590000000	N/m^2
Mass Density	1560	kg/m^3
Tensile Strength	114000000	N/m^2
Compressive Strength	122000000	N/m^2
Yield Strength	119000000	N/m^2
Thermal Expansion Coefficient		/K
Thermal Conductivity	0.2256	W/(m·K)
Specific Heat	1630	J/(kg·K)
Material Damping Ratio		N/A

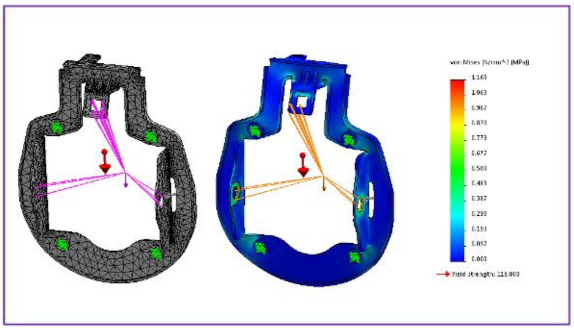


Fig. 18: Meshed part and Static Analysis.

Table 5: Result summary of existing and P0 bracket.

Analysis Results	Linear Static Analysis (Existing)	Linear Static Analysis (P0)	Delta
UY Displacement, mm (Point 1)	8.70E-07	3.03E-05	29.43E-06
Res. Max Von Misses Stress, MPa (Model)	7.929	1.160	6.769

Table 6: Natural frequencies of the initial (P0) fog-lamp bracket.

Mode No.	Frequency (Rad/sec)	Frequency (Hertz)
1	4271.5	679.83
2	4597.4	731.7
3	5288.3	841.66

4	5676.6	903.46
5	6590.3	1048.9
6	9861.1	1569.4
7	10302	1639.6
8	10353	1647.7
9	10878	1731.3
10	11732	1867.3

The first resonance frequency mode started at 679.83 Hz which is far above the operating frequency of 35 Hz. This indicator proved that even though the material changed to plastic, the structural strength of the bracket was adequate to overcome resonance from external excitation.

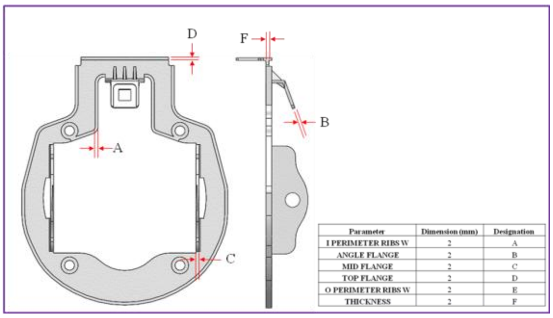
C. Optimization Process:

To minimize optimization time, the studies were split into 4 phases. Detail of all the study phases as per below:-

First Phase Optimization (P1) – The basic structure analysis was performed during this stage. All selected variables were the main construction dimension which governed the structural strength of the part.

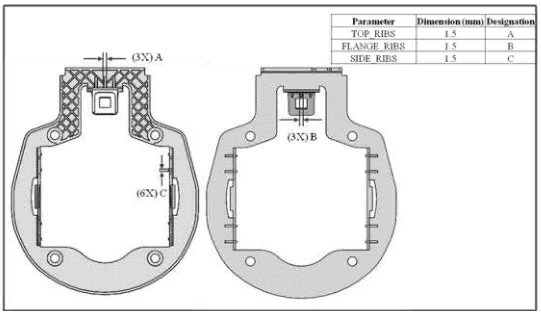
Second Phase Optimization (P2) – Ribs sizes were optimized on this phase to further increase the part strength and reduce the internal stress suffered by the model based on the linear static analysis.

Third Phase Optimization (P3) – Part weight was reduced in this phase where the focused of the reduction was on the less stress area.



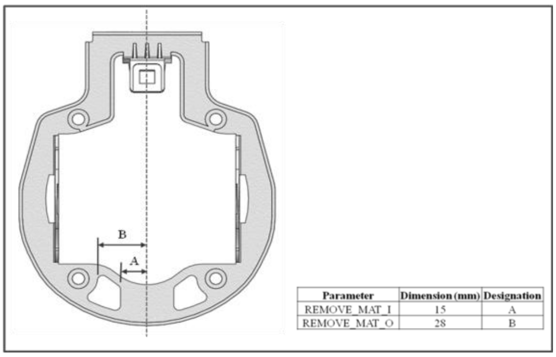
Parameter	Dimension (mm)	Designation
I PERIMETER RIBS W	2	A
ANGLE FLANGE	2	B
MID FLANGE	2	C
TOP FLANGE	2	D
O PERIMETER RIBS W	2	E
THICKNESS	2	F

Fig. 19: Parametric dimensions used as variables in the initial P1 design study.



Parameter	Dimension (mm)	Designation
TOP_RIBS	1.5	A
FLANGE_RIBS	1.5	B
SIDE_RIBS	1.5	C

Fig. 20: Parametric dimensions used as variables in the initial P2 design study.



Parameter	Dimension (mm)	Designation
REMOVE_MAT_I	15	A
REMOVE_MAT_O	28	B

Fig. 21: Parametric dimensions used as variables in the initial P3 design study.

Fourth Phase Optimization (P4) – Part was analyzed on 3 different types of materials to investigate possible weight reduction without compromising the system performance.

Data gathered from linear static and linear dynamic analyses for metal (SPCC), PBT-GF30, PP-GF30 and ABS are populated in below table for comparison purpose. The table summarized all the results from FEA analysis and information related to the materials properties and cost which are used for comparison study.

Table 7: Comparison study between existing and proposed bracket.

	Static	Dynamic	Static	Dynamic	Static	Dynamic	Static	Dynamic
Material	SPCC		PBT-GF30		PP-GF30		ABS	
URES Max Displacement (Point 1) mm	1.87E-05	2.00E-03	1.63E-05	3.00E-04	2.00E-05	5.10E-04	8.17E-05	1.60E-02
UX Max Displacement (Point 1) mm	1.23E-06	9.93E-05	4.02E-07	4.04E-05	2.19E-06	6.46E-05	5.77E-06	1.90E-03
UY Max Displacement (Point 1) mm	8.71E-07	5.77E-05	4.41E-06	1.24E-05	9.82E-06	1.83E-05	3.24E-05	1.00E-03
UZ Max Displacement (Point 1) mm	1.86E-05	2.00E-03	1.57E-05	3.00E-04	1.72E-05	5.20E-04	7.47E-05	1.59E-02
Max von Misses Stress (System) Mpa	7.81	44.08	1.59	7.66	1.61	5.60	1.60	9.07
Max Von Misses Stress (Point 2) Mpa	6.85	50.38	0.76	0.89	0.81	1.38	0.80	2.03
Max Von Misses Stress (Point 3) Mpa	5.24	20.23	0.56	2.21	0.65	0.87	0.64	3.35
Mass (grams)	140.81		39.82		28.58		26.04	
50% Yield Stress (MPa)	175		59.5		38.25		22	
1st Mode Frequency (Hz)	840.56		650.77		621.96		371.09	
Material Price Per (kg)*	0.65		1.2		1.12		2.06	
Piece Part Price (USD)	$0.09		$0.05		$0.03		$0.05	

Note*: Material cost (Plasticker.de, Accessed: 13 May 2014).

D. Summary:

Result of the comparison study in Table 7 shows that PP-GF30 was the optimum material for the fog-lamp bracket in term of part weight and cost. The maximum von Misses stress is 5.60 MPa which is below as compare to CAE stress requirement of 38.25 MPa (50% of material yield stress). In term of frequency for the first mode, it is still above the frequency requirement of 35 Hz. So both the stress and frequency conditions were satisfied by the Optimized fog-lamp which has 80% less mass as compare to existing bracket. In term of piece part cost (based on per kilogram of raw material), PP-GF30 is almost 70% cheaper than the existing metal bracket. Image below (Figure 22) shows the condition of proposed bracket in assembly condition.

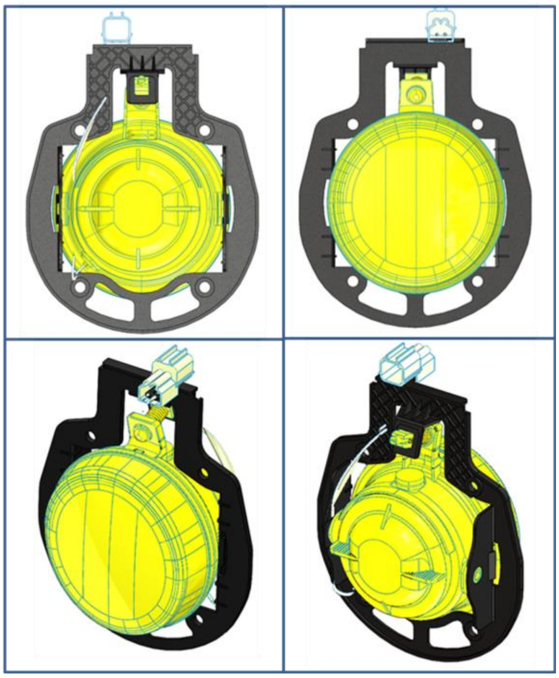


Fig. 22: Proposed PP-GF30 bracket assembled with light module.

Figure 23 shows the comparison images between existing and propose bracket; SPCC and PP-GF30. The general thickness of the bracket change from 1.2 mm to 2.0 mm. Cross ribs are added to increase strength and openings are created on the bottom area to reduce material.

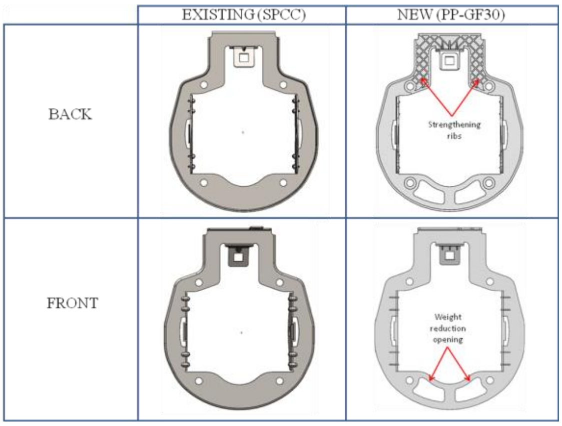


Fig. 23: Comparison between existing SPCC bracket and new PP-GF30 bracket.

6. Conclusion and Recommendation:

A. Conclusion:

The future development of Computer Aided Design and Computer Aided Engineering has provided a wide range application for the design process of automotive components. In a strict project funding, the use of Finite Element analysis is the alternative of product testing and validating and this method should be the integral part of the development of automotive components. Actual tests such as linear static, frequency analysis and linear dynamic analysis can be replaced with computer simulation and this method helped to reduce the development time and cost and by far increase product quality and reliability.

The goal of this project is to design an improved fog-lamp bracket by using computer application and simulation. This project has successful delivered the improved system based on the available current product in the market. By using product design optimisation, the performance of the existing system can be analysed and improvement can be made to the system without the use of a large scale research and development.

For this project, a conventional design process has been replaced with a hybrid process. This project have had innovated the conceptual design stage and replaced it with the product design optimisation. All iterations of products were given by the optimisation process and this helped to eliminate manual conceptualisation and concept selection process which are time consuming and complicated.

In conclusion, this research has successful produced a choice of replacement to the existing system but further refinements and validation are needed before it is ready for production.

B. Recommendation:

The full performance of the system is not being fully validated. For future works, some recommendations have been listed based on the problems in order to improve the performance.

Random Vibration:

To further evaluate the performance of the improved bracket in term of random excitation, random vibration analysis is a crucial test element. It used to calculate the response due to non-deterministic loads, such as, loads generated on the wheels of a car travelling on a rough road. Loads are described statistically by PSD (Power Spectral Density) functions. PSD function curve can be obtained from Standard MIL-STD-810F (MIL-STD-810F, 2000). The frequency domain is from 10 to 500 Hz.

Fatigue Analysis:

To examine how repeated or random load cycles can cause structural failure, fatigue analysis is recommended to be carried out. Fatigue analysis help designer's to predict component fatigue failures during development stage.

REFERENCES

Cheah, 2010. Cars on a Diet: The Material and Energy Impacts of Passenger Vehicle Weight Reduction in the U.S. Unpublished doctoral dissertation, Massachusetts Institute of Technology.

Cheah, *et al.*, 2007. Factor of Two: Halving the Fuel Consumption of New U.S. Automobiles by 2035. Laboratory for Energy and Environment, Massachusetts Institute of Technology, Cambridge.

Covill, *et al*., 2014. Parametric Finite Element Analysis of Bicycle Frame Geometries. Paper presented at the 2014 conference of the International Sports Engineering Association.

Ooi, *et al*., 2013. Parametric Optimization of the Output Shaft of a Portal Axle using Finite Element Analysis. Journal of Mechanical Engineering, 59(10): 613-619.

Mahanty, *et al*., 2001. Analysis and Weight Reduction of a Tractor’s Front Axle. Tata Consultancy Services, India, Swarup Udgata, International Auto Limited.

MIL-STD-810F, United States Military Standard, Environmental Engineering Considerations and Laboratory Tests, 2000.

Patil, Kulkarni, 2013. Optimization of Crosspiece of Washing Machine. International Journal of Research in Engineering and Technology, 2(3).

Plasticker – The Home of Plastics, "Material Exchange" in Plasticker.de. Source location: http://plasticker.de/preise/index_en.php. Available: http://plasticker.de/index_en.php. Accessed: November 07, 2014.

Sakaguchi, 2010. Influence of Diffusion of Fuel-Efficient Motor Vehicles on Gasoline Demand for Individual User Owned Passenger Cars. The Institute of Energy Economics, Japan.

Shiau, *et al*., 2009. A Structural Analysis of Vehicle Design Responses to Corporate Average Fuel Economy Policy. Transportation Research Part A, 43: 814-828.

Singh, 2008. Design an Optimized Automotive Radio Bracket Pair by Utilizing CAE Optimization Tools. Unpublished masters degree dissertation, California State University, Sacremento.

Transportation and Environment, 2013. How Clean Are Europe’s Cars? - An Analysis of Carmaker Progress Towards EU CO2 Targets in 2012. European Federation for Transport and Environment (T&E), Belgium.