Analysis and Optimization of Air Filter Unit Mounting Bracket

Mr. Ankush S. Yende[1], Prof. D. P. Kamble[2]
Department of Mechanical Engineering
[1]PG Student, Anantrao Pawar college of Engineering
[2]Professor, Anantrao Pawar college of Engineering

Abstract-*Commercial Vehicles have Pneumatic Brake System. Due to this air need to filter properly before go through whole process. Air filter mounting brackets are widely used in truck and buses brake system to hold the air filter unit with chassis. Existing mount has scope of mass optimization in current design. Finite element analysis of mounting will be done using Hypermesh and Ansys. Optistruct software will be used for topology optimization. Experimental stress of mount will be done using strain gauge and applying corresponding loading through UTM. Validation for strain vector from FEA & Experimental results will be done.*

Keywords-Air Filter Mounting Bracket, CATIA design, FEM, Harmonic vibration, Frequency.

I. INTRODUCTION

The air filter unit rest on bracket which are connected to the main-frame (chassis) of the tuck. At the time of its operation, the undesired vibrations generated by the air filter weight and uneven road surface can get directly transmitted to the bracket through the frame. This may cause damage to mounting bolts and bracket. At high frequency noise can be more. Hence, vibration removal is very much necessary in that case. For which we need to do harmonic vibration test is very much important.If the bracket have frequency which get match with engine frequency then due to large amplitude generation, may cause fatigue failure and bracket get damage. Or if the harmonic values are more than the accepted once, it may cause noise. Thus harmonic check need to do for bracket.

In today's word, it becomes very much important to use light weight material to improve fuel consumption and emission. And the most effective way to reach that goal is reducing weight. So there is strong demand in manufacturing to reduce the cost of material and increase its performance. There are no. of components are presents in vehicle which create vibration. The vibration and noise occurs because of road bumps, assembly issue, suspension results which effect on frequency band.Due to vibration present in component it may cause failure of mounting holes on chassis. The vibrations created by the road or the engine are typically at the frequencies below 30Hz. To resist all vibration created by chassis bracket must be stiff. On the other hand as the weight of air filter mounting is already mount on the bracket, due to its weight the vibration causes and it may affect the failure of bracket at the mounting holes. And if the bracket fails the unit may get fall which may cause brakes system to fail in operation.

II. LITERATURE REVIEW

Sanket C. Vinchurkar [1]- The main purpose of the engine mounting bracket is to hold the engine with chassis at any road condition. It is very much difficult to change the position of bracket and power train. This paper contains the study of Design and optimization of engine mounting bracket. For that purpose the Mahindra Scorpio Bracket is used. After analyzing the existing bracket it is found that there is scope of optimization in that bracket. For that purpose author have done CAD modeling of bracket by reverse engineering and FEA analysis on that and find the area for optimization. After find the area optimization is done.

Ajay Virmalwar [2]- This paper related with the EPB (Electronic parking brake) bracket which is from European Luxury cars. To prevent the module and the damage of bracket it is decided that the when the bracket get assembled with BIW it should have the frequency greater than 35Hz. But initially the bracket have reported the first frequency 27Hz. This value may be reduce again when the bracket get assembled on BIW. So to achieve the desired range of frequency, Optistruct software is used. Now by using topology optimization, the stiffness of bracket is get increases. After doing some corrective action it is fount that frequency gets 43Hz. And after mounting the bracket on BIW we got frequency range 36.5Hz. In that way bracket passes the safety criteria.

Tapas Sharma [3]- As we all know engine is very much important part in vehicle. It is important to keep it in proper place and position with minimum noise and vibration. Vibration and noise may cause the discomfort to the driver and passenger. There is always research goes on engine as it is main and important component in vehicle. In this paper the author did the optimization on the mounting rods and change

its cross section. For that purpose they have used three rods of material steel, aluminium and magnesium. The main purpose of using three materials rod is to reduce overall weight to increase fuel consumption and reduce emission.

Monali Deshmukh [4]- The engine mounting plays important role in reducing noise vibration weight in vehicle. As the vibration of engine is more the mounting should have damping capacity so that the vibration should not transmitted in overall vehicle. As themounting bracket is always goes in static and dynamic vibrations the harmonic vibration test is needed. Keeping this in mind, author did the static analysis, Modal analysis and harmonic analysis of bracket. For that purpose the FEA has been done on the ANSYS 15.0. First Existing model get optimized on the rib portion and the by doing FEA, Modal analysis and harmonic analysis reduced its weight. Also Vibration and noise are the main factors in consideration.

Finite Element Analysis (FEM)- For doing the FEA first the modeling of bracket is completed in CATIA software. Then by using Hypermesh we are done with Discretization. Degree of freedom is fix on mounting holes on chassis and load applied on mounting holes on unit.

a) Von-Mises-

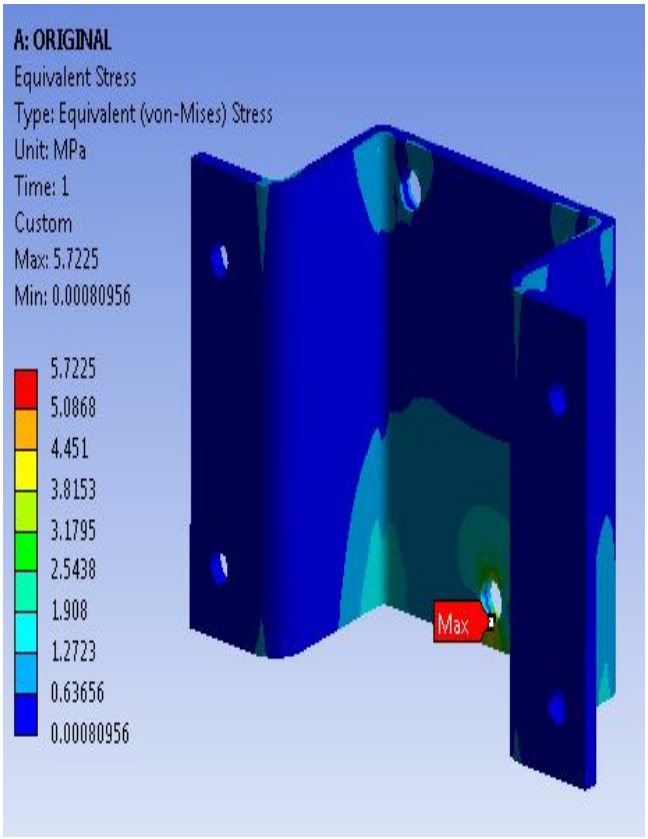

Fig 1. Von- Mises stress of air filtermounting bracket.

Maximum value of stress is 5.722 MPa.

b) Deformation-

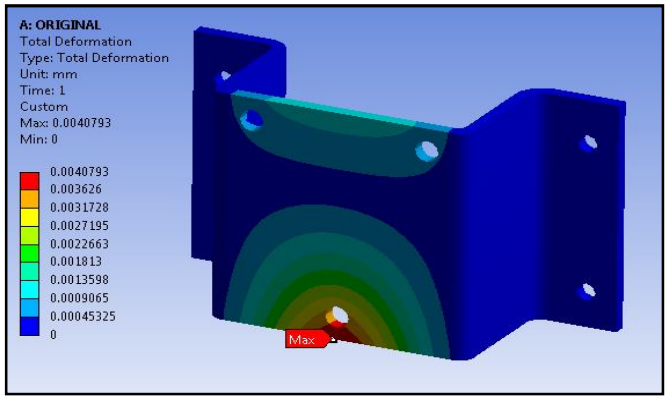

Fig 2.Deformation of air filter mounting bracket

Maximum Deformation is 0.0040 mm.

Figure shows equivalent Von-Mises stresses. Von-Mises stresses are resultant stresses or yield strength of material, after which deformation starts. Maximum value of stress is 5.722 MPa and the color plot shows the portion where the maximum amount of stress acts.

III. TOPOLOGY OPTIMIZATION

Thisoptimization is the process that optimize the material body within a design area which is given, a particular value of load and boundary conditions so that the resulting area achieve a desired goals which have set.

Steps in Topology Optimization-

The topology optimization consists of the below steps.

- Define the design area
- Define optimization parameters
- Material remove process

Optimized Model-

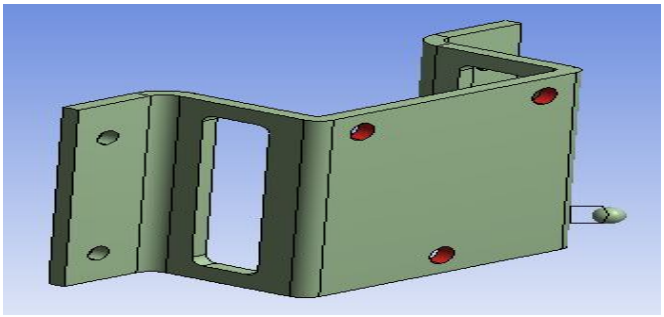

Fig 7. CAD Geometry of Optimized model

The model shown in figure 7 is optimized model. After studying the result of analysis of air filter unit mounting bracket, certain design changes have been made here. Here in this optimized model, two slots are added on the both sides of bracket. Suggested optimized model has less material and weight than previous model. Hence it results in to weight reduction of overall structure.

Boundary Conditions –

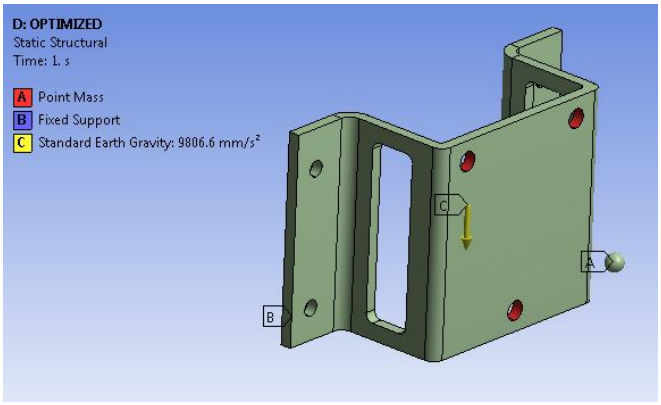

Fig 8. Boundary conditions on optimized model

To find the Von-Mises stress, Figure 8 shows the applied boundary conditions on the optimized model. Chassis Mounting holes kept fixed while the other holes put force of 8.2 kg as per filter weight.

Von-Mises-

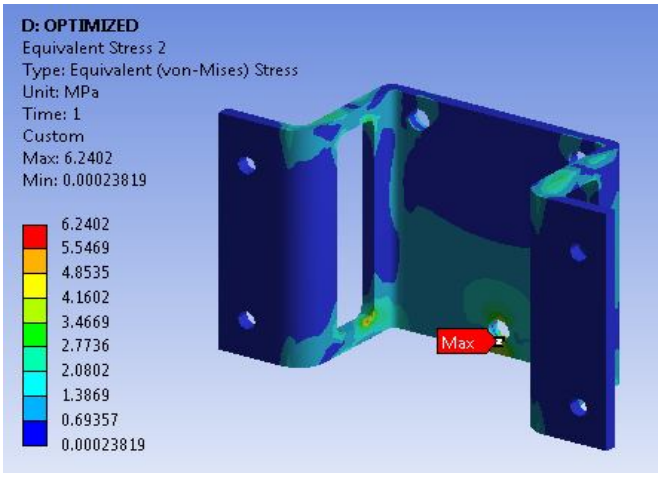

Fig 9. Von- Mises stress

Figure shows equivalent Von-Mises stresses. Von-Mises stresses are resultant stresses or yield strength of material, after which deformation starts. Maximum value of stress is 6.240MPa and the color plot shows the portion where the maximum amount of stress acts.

Deformation-

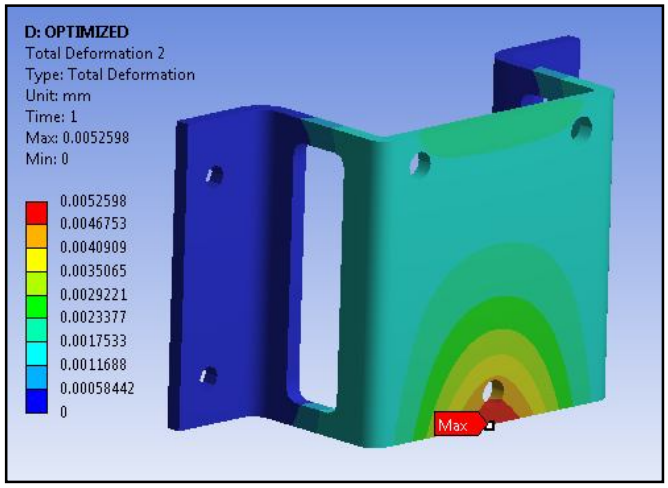

Fig 10. Deformation of Optimized model

Figure 10 shows Maximum Deformation obtained in the bracket. The maximum value of deformation is 0.0052mm. So there is slight difference between the values of existing bracket and optimized model.

IV. EXPERIMENTAL ANALYSIS

Strain Gauge Mounting Procedure-

Clean the surface of the part where you want to attach it-First use the grit paper to clear the dirt on the surface on part, then use the degreasing agent to rinsing it properly. After rinsing wipe the surface with clean cloth

Glue on the strain gauge-

This can be done by the following steps:

a. Put strain gauge on its face down on a tape.
b. Put a drop of any gluing agent on the Part.
c. Now hold the strain gauge near the superglue drop so that it get attach to the part.
d. After that apply the pressure on the strain gauge so that it will fix with part for one minute.
e. remove the tape later on.

Solder the wires to the terminals on the strain gauge-

a. Solid wire to be used for that.
b. Remove the any drop of superglue is there on terminals of the strain gauge.
c. Apply single drop of solder on the terminal of strain gauge, after that insert a with end on that drop, so that it

get stick. Then put a pressure on the wire by soldering iron so that attachment completed.

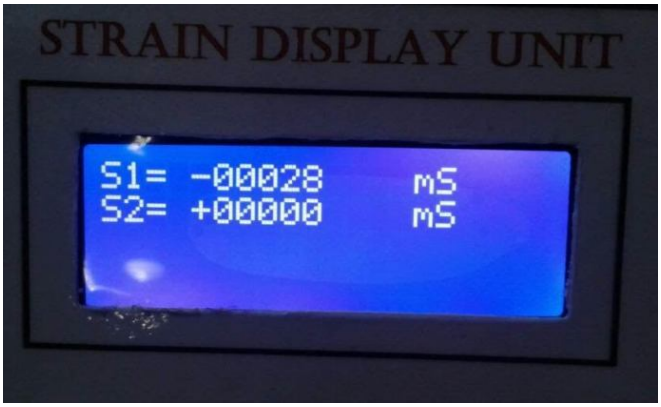

Fig 17. Strain Display Unit

Fig 18. Exprimental Setup

V. RESULT AND DISCUSSION

	FEA	EXPERIMENTAL
STRAIN	31µS	28µS

	Initial Bracket	Optimized bracket
Von-Mises stress	5.722 MPa	6.240 MPa
Deformation	0.0040 mm	0.0052 mm
Weight	3.61 Kg	2.81 Kg

Above table gives comparison between strain values what we get from FEA and by Actual Experiment.

VI. CONCLUSION

The optimization of Air filter Unit mounting bracket is attempted by applying certain changes in its design. After comparison of results obtained from analysis performed, it is concluded that the optimization attempted is found successful. And with the optimization done in the bracket it is found that as the material from the side walls of bracket is reduced the overall weight of the bracket is also reduced. The modified design of bracket is obtained which is 12.4% lighter byweight than initial on-optimized bracket. This results in material saving, and overall cost reduction

REFERENCES

[1] Sanket C. Vinchurkar and P. M. Khanwalkar, “FEA and Topology Optimization of an Engine Mounting Bracket”, International Journal of Current Engineering and Technology E-ISSN 2277 – 4106, P-ISSN 2347 – 561.

[2] B. Vijaya Ramnath, C.Elanchezhian, “Analysis and Optimization of Gating System for Commutator End Bracket”, Procedia Materials Science 6 (2014) 1312 – 1328.

[3] Ajay Virmalwar, and Pradeep Yadav, “Frequency Optimization of EPB Bracket Using OptiStruct”, Simulation Driven Innovation, HTC 2012.

[4] Jeong Woo Chang and Young Shin Lee, “Topology optimization of compressor bracket”, Journal of Mechanical Science and Technology 22 (2008) 1668~1676.

[5] Tapas Sharma, “Finite Element Analysis of Engine Mounting System”, International Journal for Scientific Research & Development, Vol. 3, Issue 06, 2015, ISSN (online): 2321-0613.

[6] Monali Deshmukh, Prof. K R Sontakke, “Analysis and Optimization of Engine Mounting Bracket”, International Journal of Scientific Engineering and Research (IJSER), ISSN (Online): 2347-3878, Volume 3 Issue 5, May 2015

[7] Yongyao Luo, Zhengwei Wang, Guodong Chen, Zujian Lin “Elimination of upper bracket resonance in extremely high head Francis hydro-generators”, Engineering Failure Analysis 16 (2009) 119–127.

[8] Michael R. Thomas, “Shape and Topology Optimization of Brackets using the Level Set Method”, Rensselaer Polytechnic Institute, Hartford, Connecticut, DECEMBER 2010.

[9] Sang EuiLee, Anil Kumar Thayamballi, Jeom Kee Paik, “Ultimate strength of steel brackets in ship structures”, Ocean Engineering101(2015)182–200.

[10] Youngwoo Choi, Dohyun Jung, Kyoungchun Ham, Sungin Bae, “A study on the accelerated vibration endurance

[11] Vtests for battery fixing bracket in electrically driven vehicles", Procedia Engineering 10 (2011) 851–856.
[12] Dean Culver, EarlDowell, "High Frequency Analysis Of A Plate Carrying A Concentrated Nonlinear Spring–Mass System", Journal of Sound and Vibration.