JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

Design Optimization of Harness Duct Fixation Bracket for improved NVH levels in LCV Engines

S.Anto Biju
Assistant Manager
Platform & Architecture

Renault Nissan Technology and Business Center India Pvt Ltd, Chengalpattu, Tamil Nadu,India 603002

Abstract : Multi Function Brackets helps to attach more than one number of parts of the vehicle like Harness Dynamic Loop Duct, Fuel Pipe & AC Pipe. The Bracket help to keep the parts on place with the Light Commercial Vehicle (LCV) Engine as vehicle experiences several torque loads and inertial loads. The mounting system with the bracket helps in controlling the motion of the engine unit, preventing it from physical contact with other components of the vehicle system from sudden impact due to vibrations. This paper investigates the multi function bracket made of steel loaded along y- direction for mounting on engine. By varying fixation of the bracket, appropriate mounting location to improve overall NVH performance of the bracket.
Keywords: Multi function brackets, mounting system, LCV Engines, NVH levels.

Introduction

A bracket is inevitable in a chassis of an automotive with regard to its functionality, which is to hold components like Harness, various ducts, pneumatic and hydraulic transmission cables etc. The primary function envisaged by these brackets is that it will keep the components at place, both in terms of orientation and position, without those being disturbed on account of various disturbing forces and torques generated during the motion of the vehicle.

Talay et. Al [1] in his research studied the optimization of Aluminum support bracket which is used to carry alternator and hydraulic steering pump. Optimization in design is done using layout analysis and thereafter, topology optimization is obtained which resulted in 37% reduction in total weight of the assembly. Another such research [2], uses FEA methods for topology optimization of brackets used for engine mounts. The model was created using CATIA V5, optimization of weight was done by considering different materials for the engine mount bracket. Cyril [3], in his paper, did modal analysis for Engine and transmission systems for positioning mounting unit and springs. Study was done for a vertically mounted V8 engine. He concluded that the overall NVH performance of powertrain can be improved by deciding the appropriate mounting location and optimizing the stiffness rates. Pavan et. al [4] did optimization on natural frequency by using different materials such as Aluminum, Magnesium and Cast iron for engine mounts using FEA and found that, Magnesium and Aluminum can be preferred over Cast iron. Irvin et. al [5] investigated the fatigue behavior of two different bracket combinations for use in ocean craft and concluded that the weld quality in the brackets contribute positively towards the fatigue loads. Akshay K and Gangdeep S M [6], performed structural analysis on bracket for Engine mounts with different materials like natural rubber, neoprene and silicon rubber. Optimization is done on the materials chosen through parameters like stress values, deflection values and natural frequency values. Analysis revealed that, silicon rubber had good vibration damping characteristics of the other two materials in the horizontal position. Shi et al. [6] did harmonic analysis on a bus body using ANSYS and studied the dynamic response parameters such as vibration transmissibility. It was found that, the second order excitation during the idle of the engine contributed much towards vibration response.

The present work focuses on a detailed study of the harness duct bracket with respect to parameters such as frequency, strength and other performance in the direction of y-axis of mounting on engine, the material of the bracket being DD13 steel (EN 10111:2008). The analysis is done using CATIA using Simulation methodology. The modelling is done in CATIA. Appropriate mount location with respect to the bracket materials chosen is analysed and checked for optimum NVH performance levels.

Methodology

A step-by-step procedure of designing the bracket and performing finite element analysis is formulated into a decision analysis matrix.

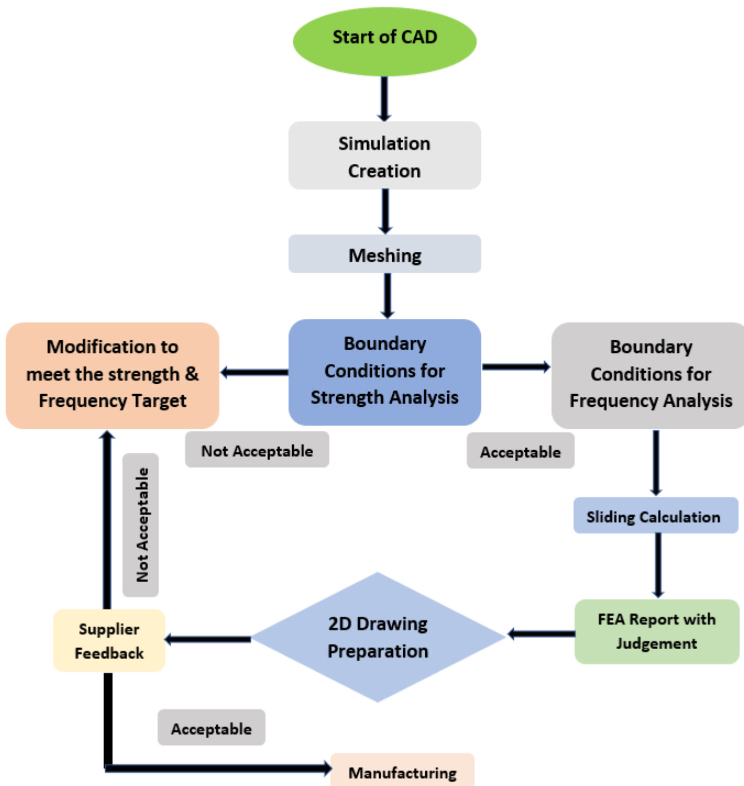

Fig. 1 Methodology

The overall work starts with a CAD model, wherein the bracket under study is modelled with the necessary dimensions and constraints. CAD model for the bracket starts from the creation of a solid volume in the available space with necessary clearance. The various parameters taken into consideration while developing the solid volume are fixation points, assembly points, fastening points etc. The final volume is created by considering these parameters.

Develop the bracket from the volume based on the standard thickness of the materials. Always start with minimum thickness and simulate the model for frequency and stress levels. This step is repeated with different values of thicknesses in increasing order till the frequency and stress levels are found to be convincing and satisfactory. The target values are chosen from standards for mounting as shown in table 1. It has to be noted that, if there is no convincing values for stress and frequency levels, then material of the bracket is changed and the simulation is repeated as described. In some cases, the stress levels and frequency values appear to be unconvincing even after having run the simulation with maximum thickness the bracket can have in a particular type of material (standard thickness available), beyond which, there cannot be any increase in the thickness values for that material. In those cases, structural features in design of the CAD model are added such as addition of ribs, welded extensions to the bracket etc., to meet the strength requirements of the bracket and the simulation is repeated as described above.

Table 1 Vibration part locations on Diesel Engines: Vibration resistance standards (34-00-037)

Gear box BV REF Overall Profile – 4 Cyl Diesel			
Frequency range (Hz)	**Vibratory amplitude/0-peak Engine system of coordinates**		
	Z	Y	X
25 – 100	25.0 µm	15.0 µm	20.0µm
100 – end H2*	100.0µm	35.0µm	50.0µm
End H2 – end H4*	2, 5 G	1, 5 G	2, 5 G
340 – 500	2, 5G	2, 0 G	1, 5 G

*Wherein End H2 refers to 187 Hz and End H4 refers to 374 Hz.

The finalization of the design is done once the simulation results for frequency and stress analysis are well within the specified range for that particular material which is considered to be satisfactory for the bracket. Thereafter, a manufacturing feasibility study is done through cross domain with the suppliers and manufacturing engineers, post which, the pilot batch production of the bracket is initiated.

The current work focusses on the modelling and analysis of the bracket with DD13 steel and finalizing the dimensions and loading points on the bracket. CAD modelling is done using CATIA and analysis is done using ANSYS software. The manufacturing feasibility study is beyond the scope of the current work and hence, the same is not briefed in this paper. The methodology shown in Fig. 1 contains both design and manufacturing phases. The scope of current paper stops after FEA report. The properties of DD13 steel is as shown in Table. 2.

Table 2 Properties of DD13 steel

Property	**Specification/values**
Standard	EN 10111: 2008 Continuously hot rolled low carbon steel sheet and strip for cold forming.
Chemical composition	0.08%C, 0.4% Mn, 0.03%P, 0.03%S.
Tensile strength	400 MPa.
Yield strength	170-330 MPa for 1-2 mm thickness

Results and discussion

The bracket is modelled with thickness of 2 mm and subject to meshing and analysis of stress and frequency. The induced frequency is found to be 53.867 Hz which is way less than the expected target frequency of 187 Hz. The induced stress is found to be 517 MPa. The Von-Mises stress distribution for the bracket with 2 mm thickness is as shown in Fig. 3.

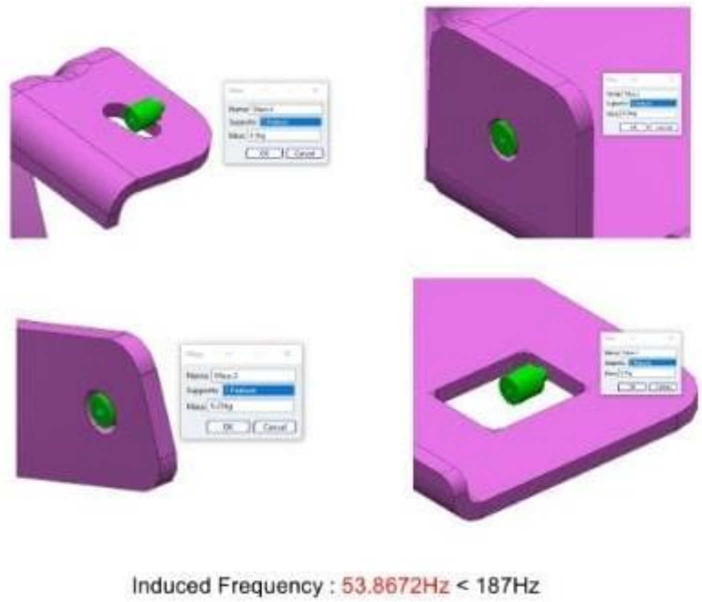

Fig. 2 Frequency analysis of bracket with 2 mm thickness

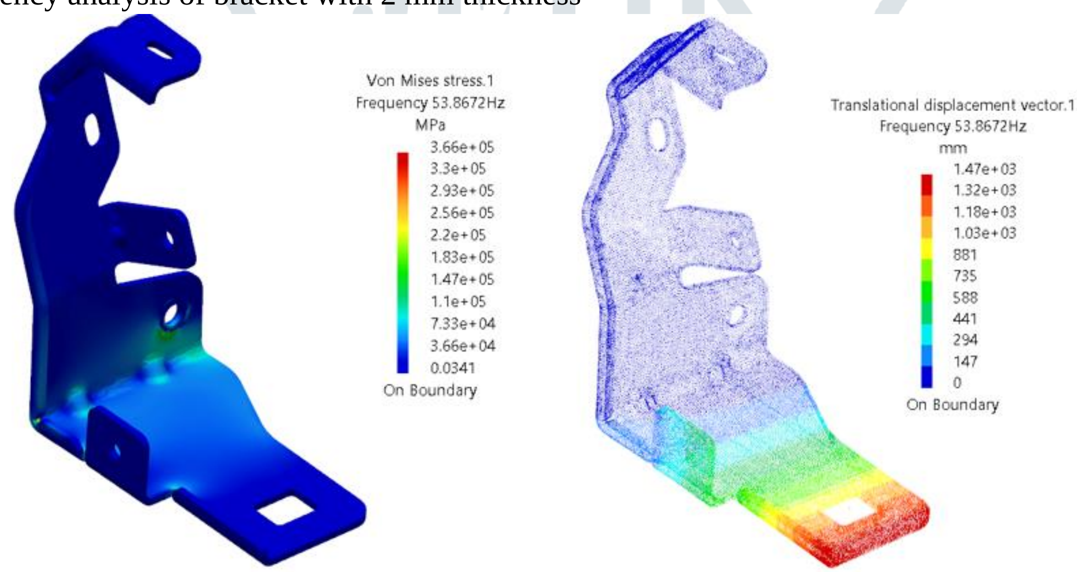

Fig. 3 Stress analysis of bracket with 2 mm thickness

From Figures 2 and 3, it is understood that, the stress and frequency values obtained against 2 mm thickness of the bracket is not satisfactory. And hence, the thickness value is increased by 1 mm i.e., CAD model of the bracket is rebuild with new thickness of 3 mm and the simulation is repeated to obtain the stress and frequency values, which is found to 380 MPa induced stress and 119.50 Hz modal frequency. Again, the values are not quite convincing when compared with the standard values. Fig. 4 and 5 shows the modal frequency and stress distribution in the bracket of 3 mm thickness.

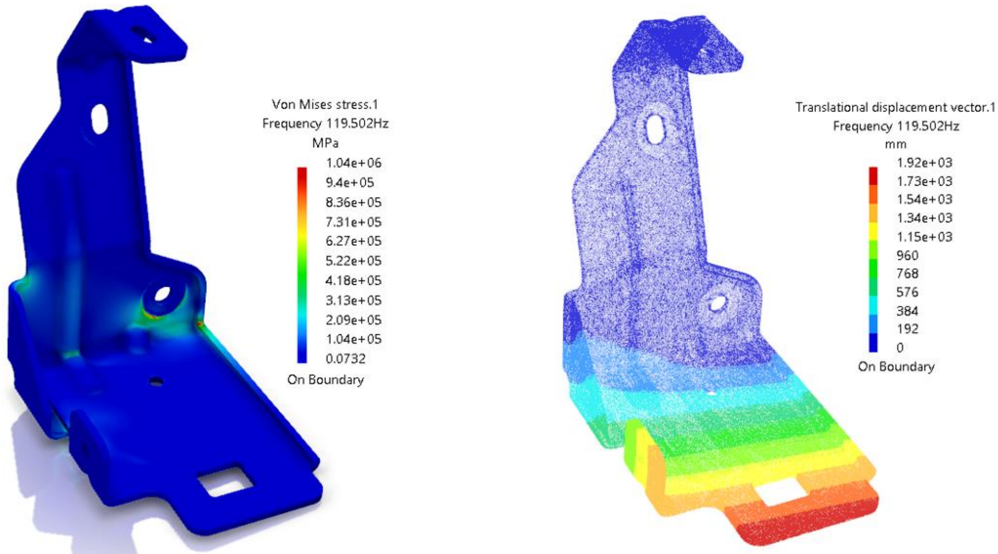

Fig. 4 Modal frequency of bracket with 3 mm thickness

Now, the maximum thickness permitted for the selected material for bracket (DD13 steel) is only 3 mm. But, it is evident from the modal and stress analysis that, the frequency and stress levels obtained are not within the permissible range. Hence, as discussed in the section 2 of this paper, structural modifications to the bracket is done by adding ribs to it.

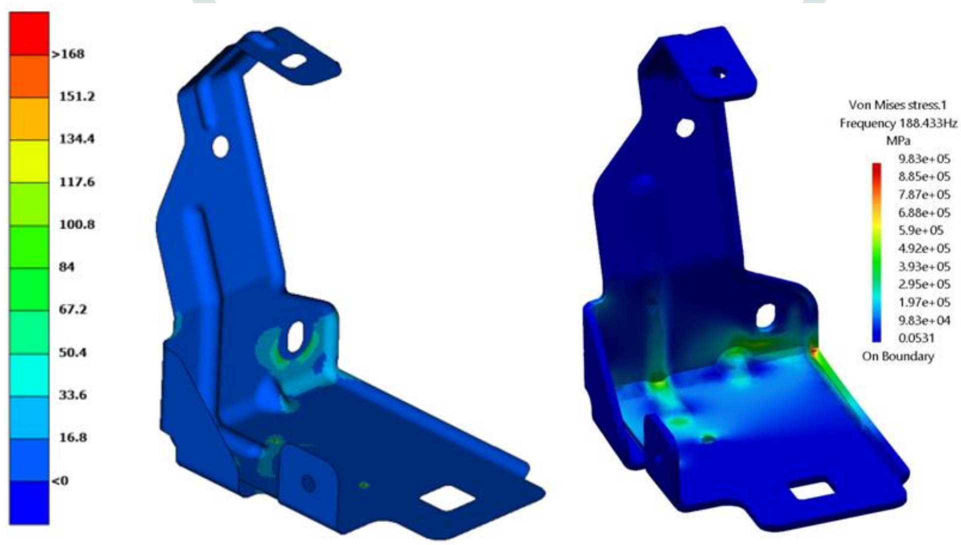

Fig. 5 shows the bracket with structural additions like ribs added to it.

The maximum von-mises induced stress is found to be 168 MPa, with a displacement of 0.12 mm which is comparatively less than the former two brackets (Fig. 3 and Fig. 4). Also, it has to be noted that, the modal analysis result was quite convincing with an obtained value of frequency as 188 Hz.

The values of maximum induced stress, displacements and modal frequencies obtained in various design iterations of the bracket are as tabulated in table 2.

Table 2 Design iterations and parameter values

Iteration number	Bracket thickness (mm)	Von-Mises stress (MPa)	Displacement (mm)	Obtained Frequency (Hz)	Expected Frequency (Hz)	% deviation
1	2	517	2.1	53.867	187	71.19%
2	3	380	0.8	119.50	187	36.09%
3*	3	168	0.12	188.43	187	0 %

*structurally modified bracket.

Conclusion

The present work focused on the effect of structural topology of the bracket made of DD13 steel, on the induced stress values and modal frequencies. The thickness chosen were 2 mm and 3 mm as per the standard dimensions of the bracket materials and hence beyond the maximum standard thickness, the analysis is not carried out as it is found to be not necessary. From the literatures, the target value for modal frequency is set at 187 HZ and the frequencies obtained in the modelling trials were compared against the target frequency in such a way that, minimum the deviation from the target value, better the performance of bracket in NVH. It is found that, even though by increasing the thickness to 3 mm yielded good result with respect to the stress induced in the bracket for the given load., the deviation with respect to modal frequency was more (36.09%), but better against a thickness of 2 mm, which yielded a modal frequency of 53.867 Hz (71.19% deviation). The structurally modified bracket (by keeping thickness same at 3 mm) yielded a maximum stress of 168 MPa, which is less when compared against 517 MPa and 380 MPa, for structurally unmodified bracket with thicknesses of 2 mm and 3 mm respectively. The deviation from the target frequency was found OK, which makes the structurally modified bracket with 3 mm thickness more suitable for the application specified.

References

1. Talay, Erkan, Cengiz Özkan, and Emre Gürtaş. "Designing lightweight diesel engine alternator support bracket with topology optimization methodology." *Structural And Multidisciplinary Optimization* 63 (2021): 2509-2529.
2. Kirthana, S., and Mohammed Khaja Nizamuddin. "Finite element analysis and topology optimization of engine mounting bracket." *Materials Today: Proceedings* 5.9 (2018): 19277-19283.
3. Cyril, S., and H. Manikandan. "Optimization of engine and transmission mounting system for a V6 front wheel drive vehicle." *Materials Today: Proceedings* 72 (2023): 2564-2568.
4. Pavan B. Chaudhari and Dr. D. R. Panchagade "Comparison of Magnesium, Aluminium and Cast Iron to obtain Optimum Frequency for Engine Bracket using Finite Element Analysis", IJERA Vol. 2, (2012), pp1016-1020.
5. S. Irving , F.Ferguson-Smith, X.Z. Hu, Y. Liu, "Comparative fatigue assessment of soft toe and nested bracket welded aluminium structures", Engineering Failure Analysis 12 (2005) 679–690.
6. Akshay Kumar, Gangdeep Singh Mavi, "Analysis of Engine Mounting Bracket for an Automobile using FEA", **Indian Journal of Science and Technology,** *Vol 11(28)*
7. Shi Q, Wang T, Li H. Harmonic response analysis and research of bus body based on the modal shapes. International Journal of Vehicle Noise and Vibration. 2011; 7(3):253–70. https://doi.org/10.1504/IJVNV.2011.042423