Automobile components of vehicle under controlled dynamic conditions

[1]Rajiv Khullar, [2]Sachin Kalsi
[1,2]Assistant Professor, ME,, Chandigarh University, Mohali,, Punjab, India
rajk.libra@gmail.com, sachin.uie@cumail.in

Abstract
With the increase of competition in the field of automobile air conditioning industries, they are looking for quality, cost effectiveness, short time to market and easy way to solve their problems. Therefore, a simulation need is at a high level. In the present work, dynamic analysis of automobile air conditioning system mounting brackets like compressor mounting bracket is proposed to analyse the condition of bracket under various vibration loads as per industrial standards. Then, new optimized design of bracket using different structural optimization techniques are to be proposed with the given constraints.

Keywords—optimization;compressor mounting; dynamic; analysis; finite element analysis; hypermesh; solidworks.

I. INTRODUCTION

For a designer team or a company, the design and development of an automobile is a big and complex task to fulfill all the requirements of a vehicle according to customer need and different standards adopted to authorize the production of that automobile. But, an automobile is an extremely market- oriented commodity. So, with instantaneous and regular feedback from the public, constant and consistent development is possible. So, the customer wants maximum comfort for minimum price. So, competitors in this field are always ready to give best to customer in lower price. To solve these problems, they always try to minimize labour, raw material cost and other costs that are required to manufacture an automobile. So, they are using computer aided engineering as one of the method for solve these problems in a short and effective time.

II. LITERATURE REVIEW

Many researchers have worked in the field of vibration analysis of many components using finite element analysis approach. **Chang and Lee [1]** performed work on compressor bracket and used topology optimization. With the adjacent vehicle parts, base bracket model by considering the interference for the topology optimization was modeled. **Chaudhari and Panchagade[2]** using finite element analysis approach, different materials are compared which are aluminum, magnesium and cast iron to obtain optimized natural frequency for engine bracket. **Choi *et al.* [3]** done a study on vehicles which are electrically driven in which the battery fixing bracket is mounted to accelerated vibration endurance tests. **Fernando M.D. Ramos [4]** performed the vibration analysis of a mount of an engine. Created a finite element model of an automotive elastomeric engine mount to study the dynamics of a component. **Gagandeepsingh [5]** designed an optimized automotive bracket of radio using CAE optimization tools. Altair software, Hypermesh was used for finite element model and Optistruct was used for finite linear and dynamic analysis. **ZhijiaZ.Yang [6]** performed vibration analysis of bracket of motor used in automotive steering columns to determine the structural integrity when subjected to vibration.

Two most important criteria which are required for the engine mounting bracket are:

Firstly highly damped and stiff engine mounting brackets were used in order to control the vibrations for road induced and idle shake. Secondly for acoustic comfort and vibration isolation, a mounting bracket of lightly damped and a compliant for the higher frequency range (30-250Hz) is required [7]. BMW's 8-cylinder power units 43 use magnesium Inlet manifolds fully variable and also Volkswagen has a inlet manifold of magnesium for the Phaeton [8]. For controlling movement of powertrain and vibration due to dampening powertrain relative to the vehicle and to maintain the transmission mounting brackets in the transmission isolators related to the support members are arranged [9].

The design of bracket is optimized by the use of computer aided engineering for mass, stiffness and strength. The stiffness of bracket must be taken into consideration because for the maximum isolation of vibration is desired for the rubber to be soft [10].

Higher strength and stiffness and in the same time reduce weight is desirable while vehicle structure design. In design, various types of optimization methods have used in the field of structure design of vehicle [11] [12]. In general, the size optimization method as design parameters aims to optimize properties such as section modulus, poison ratio, Young's modulus, beam area, thickness of plate, density, etc [13]. The topology optimization is a method to remove unnecessary materials from the current volume and in the initial design phase to determine a fundamental shape as well. In various load conditions, its ability to identify which portion of the current model is necessary to remain to form a final shape is the advantage of this method [14] [15].

III. CAD MODEL AND MESHING OF BRACKET

A. SHAPE OF BRACKET AND ITS CAD MODEL

The Bracket was borrowed from market of passenger vehicle and reverse engineering was performed to obtain dimensions of a bracket, so that its CAD model can be generated as shown in Fig. 1 and 2..

a.) Front View b.) Rear View

Fig. 1 (a) & (b): Compressor mounting bracket

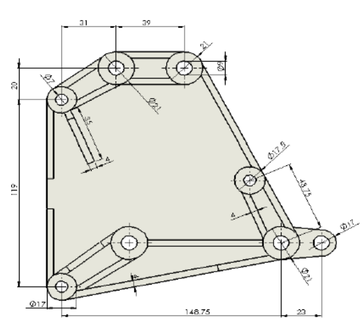
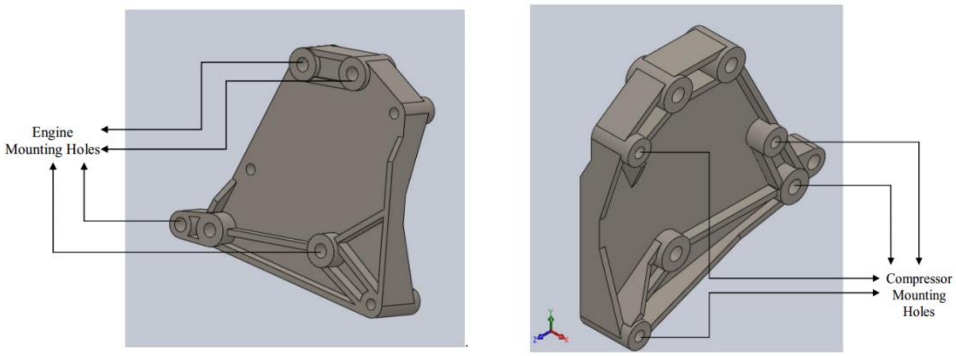


Fig. 2: Dimensions of bracket after reverse engineering

From the given dimensions , the bracket is modeled in CREO software. The CAD model with the constraints are shown in Fig. 3.



(a)(b)

Fig. 3: Mounting holes on bracket

B. Meshing of compressor bracket

The meshing of bracket was done using Altair Hypermesh software and the element type used for this purpose is tetrahedral as shown in Fig. 4.

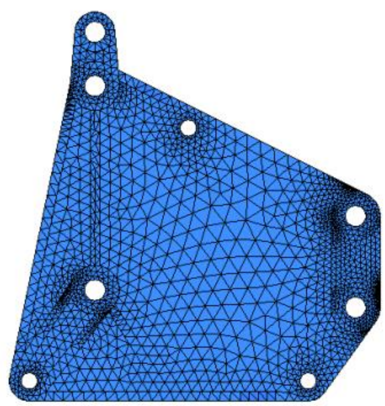


Fig. 4: Tetrahedral meshing of bracket

IV. DYNAMIC ANALYSIS OF BRACKET

The dynamic analysis for the compressor mounting bracket of passenger vehicle is done in two steps.

A. Normal modes analysis

To find mode shapes and the natural frequencies of the bracket, this analysis was done. The material chosen for this analysis was aluminum.

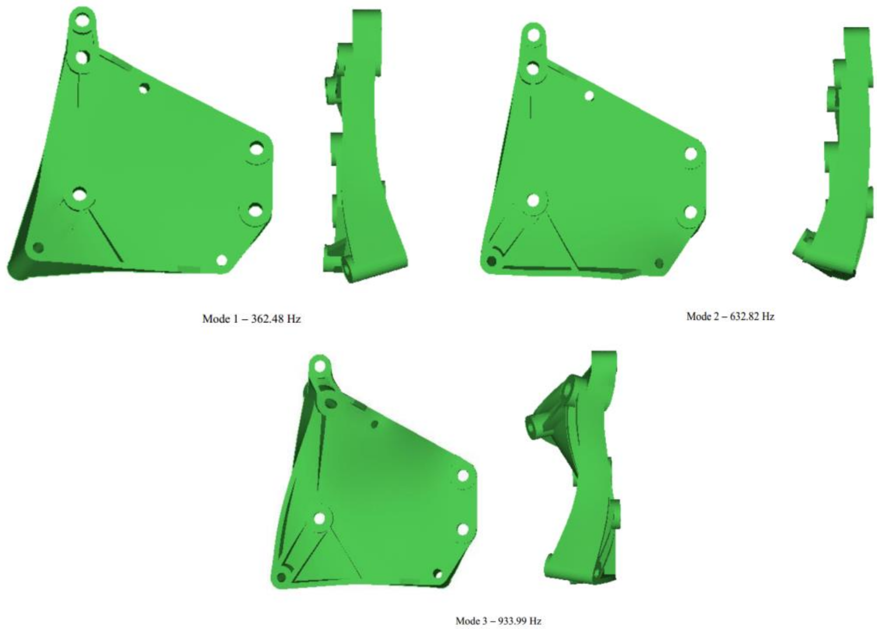


Fig. 4: Different modes of bracket

The natural frequencies with their corresponding different mode shapes can be seen from the Fig. 4.

B. Modal frequency response analysis

An external harmonic excitation in the modal frequency response analysis is given by the external force. Modal frequency response analyses of compressor mounting bracket along all three axes are done. CAD data and meshing has been explained in the previous sections and is same for all the analyses of the compressor mounting bracket.

1)*Modal frequency response analysis along x-axis*

In this analysis, force is applied in the x-axis direction and compressor is free to move in this direction. Forces are applied according to industry standards (JIS1601D) for the vibration testing of the automobile air conditioning equipment. The von misses stresses produced can be seen from Fig. 5.

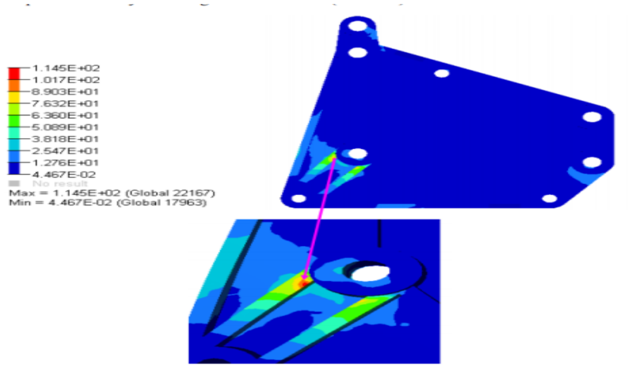


Fig. 5: Von misses stress in X axis

2) Modal frequency response analysis along y-axis

In this analysis force is applied in the y-axis direction and compressor is free to move in this direction. Except first two load collectors; all boundary conditions are applied in same way as for modal frequency response analysis along x-axis. The von misses stresses produced can be seen from Fig. 6.

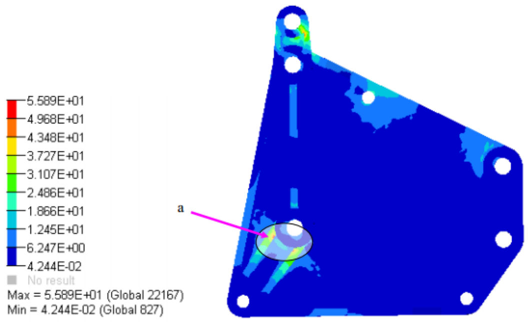


Fig. 6: Von misses stress in Y axis

3) Modal frequency response analysis along z -axis

In this analysis, force is applied in the z-axis direction and compressor is free to move in this direction. Except first two load collectors; all boundary conditions are applied in same way as for modal frequency response analysis along x- axis. The von misses stresses produced can be seen from Fig. 7.

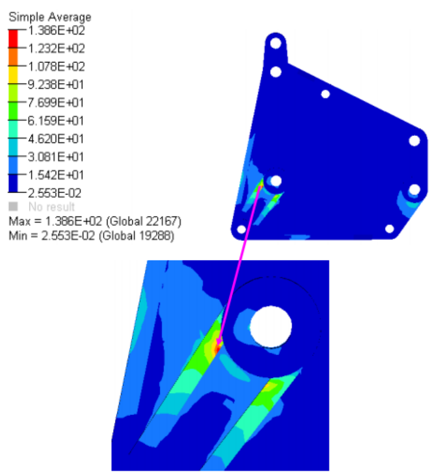


Fig. 7: Von misses stress in Z axis

V. OPTIMIZATION OF COMPRESSOR MOUNTING BRACKET

A. CAD MODEL OBTAINED BY REMOVING OF MATERIAL

The material was removed from the low stress region of the bracket as shown in Fig. 6 .CAD model was meshed and the methodology proposed for the dynamics analysis was used on the modified bracket.

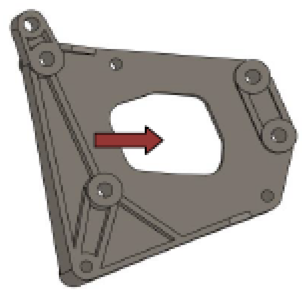


Fig. 8: Modified mounting bracket

Tetrahedral element meshing of the bracket is shown in Fig. 9.

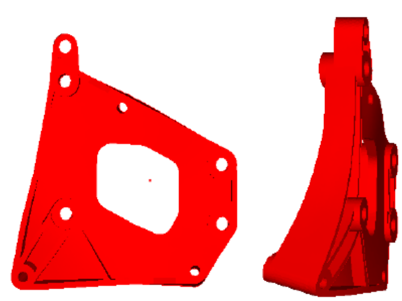
Fig 9: Tetrahedral meshing of the modified mounting bracket

B. DYNAMIC ANALYSIS OF MODIFIED BRAKCET

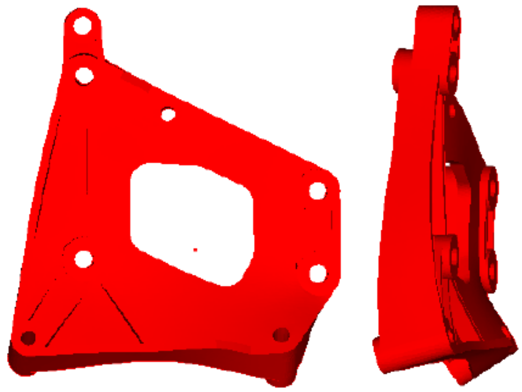
Dynamic analysis for the compressor mounting bracket is done using Normal modes analysis.

Normal Modes Analysis of the Mounting Bracket

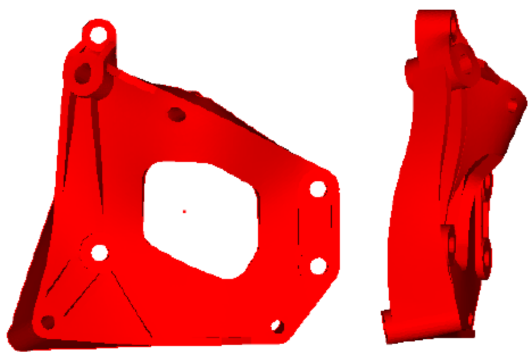
The procedures of normal modes analysis of modified mounting bracket are same as explained earlier. The results obtained are:



Mode 1 – 359.91 Hz



Mode 2 – 625.44 Hz



Mode 3 – 914.41 Hz

Fig 10: Natural frequencies and mode shapes

VI. CONCLUSION

.

In order to detect the problems in the cycle of design, the data comparison and visualization is helpful by the use of CAE tools. Simulation techniques are incorporated to save the time and for design iterations. In this paper, the mass of the bracket is reduced by about 9.4%. For the same amount of applied load, excitation frequency equivalent with the natural frequency for the stress has been observed.

REFERENCES

[1]Jeong Woo Chang and Young Shin Lee, (2008)," Topology optimization of compressor bracket", Journal of Mechanical Science and Technology, Vol. 22, pp.1668-1676

[2] Pavan B. Chaudhari and Dr.D. R. Panchagade, (2012)," Comparison of Magnesium, Aluminium and Cast Iron to obtain Optimum Frequency for Engine Bracket using Finite Element Analysis", International Journal of Engineering Research and Applications (IJERA), Vol. 2, Issue 5, pp.1016-1020.

[3] Youngwoo Choi, Dohyun Jung , Kyoungchun Ham and SunginBae, (2011), "A study on the accelerated vibration endurance tests for battery fixing bracket in electrically driven vehicles", Procedia Engineering, Vol. 10, pp. 851-856

[4] FernandoM.D.Ramos, (2008),"Vibration Analysis Of An Engine Mount", Thesis of master's degree in mechanical engineering, Instituto Superior TecnicoUniversidadeTecnica De Lisboa.

[5] Gagandeep Singh, (2012), "Design An Optimized Automotive Radio Bracket Pair By Utilizing CAE Optimization Tools", Masters of science thesis in Mechanical Engineering, Chalifornia State University, Sacramento.

[6] Zhijia Z. Yang, (2002),"Vibration Analysis of Motor Bracket Used in Steering", ABAQUS Users' Conference.

[7] Pan X., 'Structural Optimization for Engine Mount Bracket', SAE Technical Paper Series 2007-01-2419

[8] C. Blawert, N. Hort and K.U. Kainer, 'Automotive applications of Magnesium and its alloys', Transaction Indian Inst. Met. Vol.57, No. 4, August 2004, pp. 397-408.

[9] Igami H., Thompson M., Osodo K., 'Automotive Powertrain Mounting System Design Optimization', SAE Technical Paper Series 2003-01-1667.

[10] Chang H., 'A Study on the Analysis Method for Optimizing Mounting Brackets', SAE technical paper series 2006-01-1480.

[11] Y. Jeong, K. Back, W. Cho, " Multi-Disciplinary Vehicle Styling Optimization", SAE, 2003.

[12] N. Bakhtiary, "A New Approach for Sizing, Shape and Topology Optimization", SAE, 1996.

[13] C. Birkett, K. Ogilvie, Y. Song, "Optimization Applications for Cast Structures", SAE, 1998.

[14] R. J. Yang, T. J. Walsh, P. A. Schilke, "A General Formulation for Topology Optimization", SAE, 1994.

[15] Altair Engineering, "Optimization, DOE, Stochastic Study Using HyperWorks"