Study of Design and Analysis of Air Conditioner Compressor Mounting Bracket

Vyankatesh D. Pawade[1], Pushkaraj D. Sonawane[2]

[1]Student ME Mechanical Design Engineering, Maharashtra Institute of Technology, Kothrud, Pune, Maharashtra, India

[2]Assistant Professor ,Department of Mechanical engineering, Maharashtra Institute of Technology, Kothrud, Pune, Maharashtra, India

Abstract: ***Parameters like cost of vehicle and fuel efficiency are mostly influenced by the weight of the vehicle in the automotive industries. As per the safety standards this is very important to design the light weight component. This paper describes the study of the optimized design of the Air-Conditioner compressor mounting bracket. The study of the topology optimization is done as per the requirement of the bracket design. This study also highlights the factors for the failure of the mounting bracket and the effect of the optimization by various analysis.***

Keywords: Structural Topology Optimization, Mounting Bracket, Natural frequency, FEM analysis, Design and non design domain

1. Introduction

While designing the vehicle structure it is very tough job to obtain the higher stiffness and strength and also minimize the weight of the component. Compressor mounting bracket is the bracket used to mount the air conditioner compressor in the car. Mounting bracket goes under certain problems like design space issue, material used , weight of the bracket affecting the performance etc

I. Types of brackets

A. Engine Mounting Bracket of Car

Engine mounting bracket of the car is the bracket used to mount the engine from the back side. It is made of steel. The large face of the bracket is connected to the engine while the small end of the bracket is connected to the vehicle structure for taking load and vibrations. Due to less vibration rate and knocking rate of the engine its operational life is more. But if the engine is old or there are some other problems related with the vehicle structure, then there are large chances of failure of the engine mounting bracket. Crack in the bracket is the main failure due to high stresses generated in the bracket.

Figure 1: Engine mounting bracket of a car[1]

B. Aeroplane engine's continental engine mounting Bracket

A mounting bracket is used as a base member having a flat upper surface and an elongated shoulder extending upward from the base surface. The mounting bracket consist of bracket member having an upper surface adapted to support a component and a flat lower bracket surface. The base is connected to the plane structure and the other part connected to the engine which takes most of the load. It is made up of aluminum casting.

Figure 2: Aeroplane engine's continental engine mounting Bracket[1]

C. AC compressor mounting bracket

The compressor plays a very important role in the automotive air conditioning system. The unbalanced forces produced from the engine and compressor causes the structure vibrations. The compressor is supported by the engine mounting to reduce the vibratory forces is called compressor mounting bracket.

Figure 3: Compressor mounting bracket [2]

2. Related Work

G. Phani Sowjanya[3] performed a study on vibration parameters to test the avionic equipment. Vibration is the most important failure of in avionic equipment. The avionic equipment fitted into the aircraft undergoes high vibrations. A fixture was placed between the equipment and the machine. The frequency level of the aircraft as per military standards range from 20Hz to 2000Hz. An effective design of the fixture is required for proper transmission of input. It was analyzed by using finite element analysis, after selecting the suitable design of vibration fixture.

Pavan B. Chaudhari.[4], optimized the natural frequency of engine mount bracket by using three different lightweight materials by using finite element analysis. Selected materials were Aluminium, Magnesium and Cast Iron . Evaluation of proposed model of engine mount bracket was performed using finite element analysis (FEA) and modal analysis techniques. It conclude that Mg alloy had higher natural frequency followed by Al alloy. Traditional impulse and periodic impulse method were used to measure the natural frequency.

Jeong Woo Chang[5], Optimize the topology at the concept design stage where structural analysis methodology of compressor bracket was verified on the static and dynamic loading condition. New bracket shape on the topology optimize results which compared with the traditional concept model. It analyzed that a new bracket would not fail during a vibration testing and these results were verified with a fabricated real sample under the durability condition

A.Vaidya, S. Yang and J. St. Ville and Marco Cavazzuti and Luca Splendi [6]in their technical papers have defined and compared various types of Optimization techniques like topology, topography, topometry, size and shape optimization. They have presented advantages and disadvantages of various types of optimization which assisted us in determining the type of optimization to be implemented.

Erke Wang[7] presents some analytical results and some test results for different mechanical problems, tetrahedral and hexahedral shaped elements were used by simulated finite element analysis. The paper showed the comparison for linear static problems, modal analysis and nonlinear analysis.

3. Steps in Design of Bracket

1) Determine the centre of gravity of location of engine: Centre of gravity is the point at which the body acts as if its whole mass is concentrated at that point. During analysis of forces CG proves very important.

2) Selecting the proper location for individual component: The proper location of the individual component of bracket like ribs, engine mounting holes, compressor mounting holes etc, is to be selected as per the design and non design space.

3) Selecting the coefficient of stiffness for every mount: The coefficient of stiffness for the every mount should be selected which would help to know the resistance to the deformation of the component.
4) Applying boundary condition to the mount: The proper boundary conditions should be applied as per the forces acting over the mount.
5) Bracket Optimization: The optimum design of the bracket is generated through optimization.
6) Final CAD model of the bracket: As per the design obtain after optimization the final CAD model is developed.

4. Topology Optimization

At the design stage the concept of the topology optimization is very important. It is common habit to design, depending on the designer's experience at the early stage of product development. Reliable and satisfactory results with the verified structural model is obtained by topology optimization. Topology optimization is a method which distributes the density of an initially homogenous volume to achieve a certain objective function while observing the defined constraints. The main objective function is minimizing volume and the displacement acts as a constraint and with manufacturing constraint such as casting of the bracket.

Initially we have to collect the information regarding different loads acting on the bracket. The base bracket results from testing and finite element analysis (FEA) point of view for evaluating final optimized design. A structural domain consists of many rectangular perforate materials in the structural optimization topology and these microstructures within design domain material are reproduced to maximize structural stiffness.

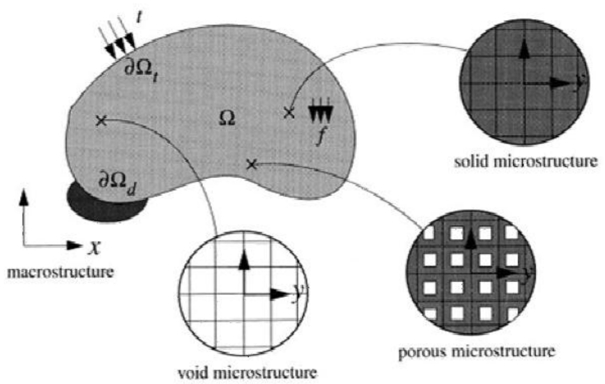

Figure 4: Design domain and microstructure [8]

Total optimization process has following steps

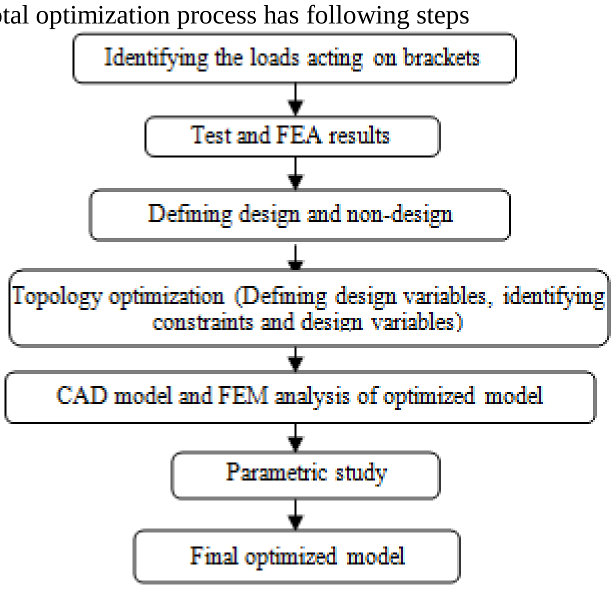

Figure 5: Flow chart for optimization

5. FEM Analysis of Mounting Bracket

Finite element analysis is one of the best suited method for the nonlinear analysis of the engineering problems. As a geometric input finite element mesh is required for the FEA. The mesh can be directly generated from the CAD model. Since CAD model is a complex geometry some changes is to be made, such as changing material, changing mesh size for proper results.

Various mechanical related FEA software are involved in this processes. The software such as AUTOCAD, PRO-E, SOILDEDGE etc are mostly used for the CAD modeling of the component and the software like HYPERMESH, ANSYS, RADIOSS, OPTISTUCT are used for meshing and various analysis.

6. Case Study for Structural Topology Optimization of Compressor Mounting Bracket [6]

Following steps are for the topology optimization of mounting bracket:
(1) Defining the design space
(2) Defining optimization parameter
(3) Processes for material removal and design details

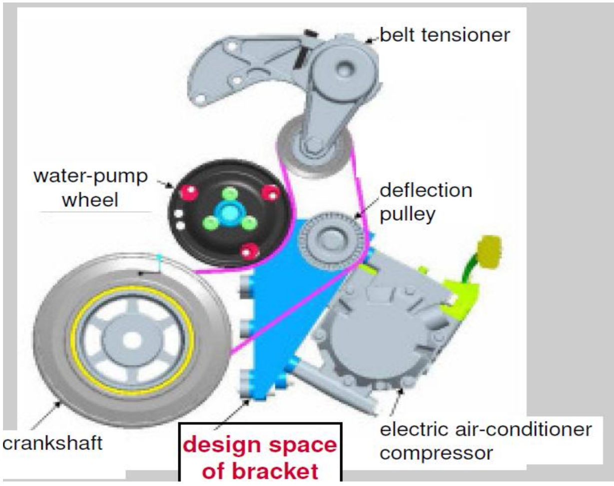

Figure 6: Design space for mounting bracket (volkswagen)[6]

Component when being held in steady condition or in working condition the a space which is available does not interfere with any surrounding components is called the design space of the mounting bracket.

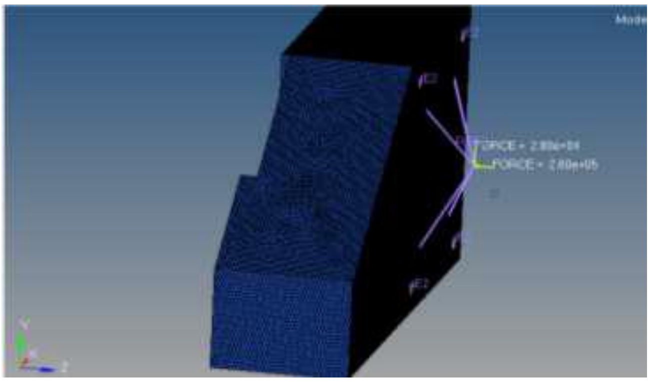

Figure 7: Forces [6]

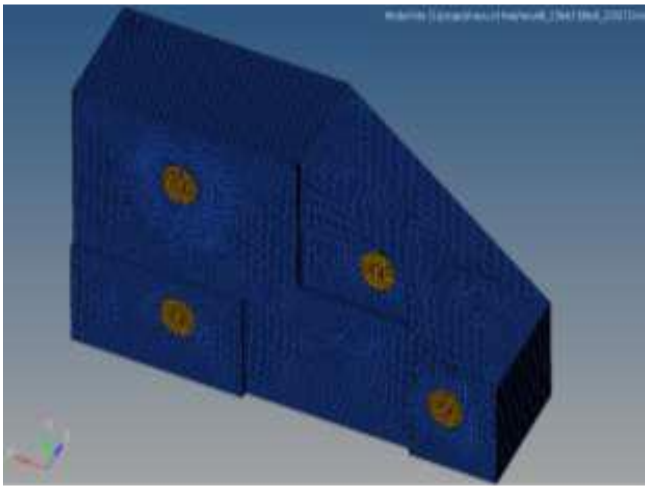

Figure 8: Design space for mounting bracket[6]

A. Boundary Condition
The forces and the constraints which are experienced by the bracket were applied as the boundary condition
The forces are,
(1) Considering the dynamic condition the weight of the bracket is 280N[6]
(2) Considering the dynamic condition belt tension is 2800N[6]

B. Defining the Optimization Parameters
Topology optimization mainly focus on minimizing the volume without affecting the strength of the bracket and the bracket stiffness. Is the parameter defined as constraint Maximum allowable displacement of bracket.

C. Process for material removal and detail design
Removal of the unnecessary material from the design space 30 iterations were required[6].
Conceptual CAD model is made from the optimized material distribution.

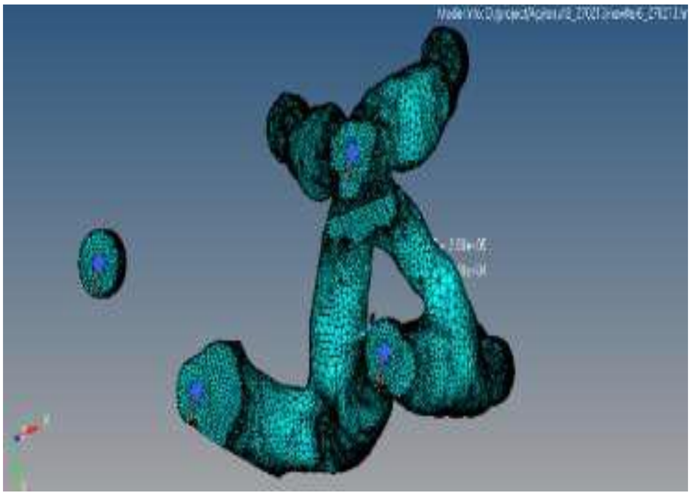

Figure 9: Optimized material distribution [6]

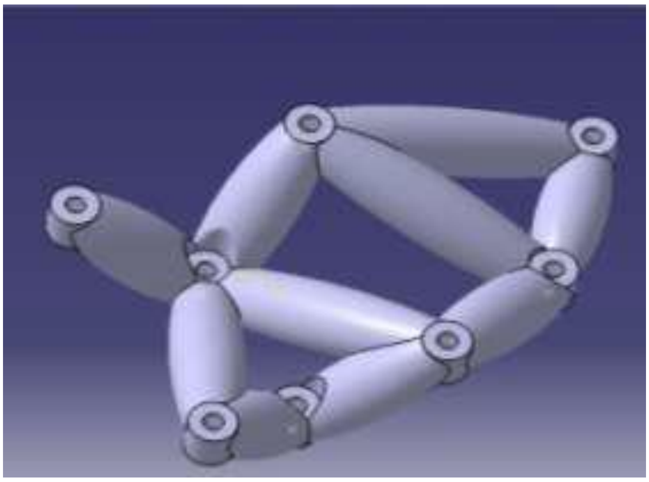

Figure 10: Optimized CAD model [6]

From the structural topology optimization process we can see that the mounting brackets weight reduction is done without affecting its strength and stiffness parameters. The process gave the optimized CAD model of the mounting bracket considering the design space for the mounting bracket.

7. Case Study for Finite Element Analysis and Natural Frequency of Engine Bracket[1]

The existing design has 4 holes. One hole is fixed and remaining three have force of 1000 N. This force is produced by Thrust. There is also self weight (g). The material used for FE Analysis is Non Linear. The FEM Model having 6 freedoms: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes.

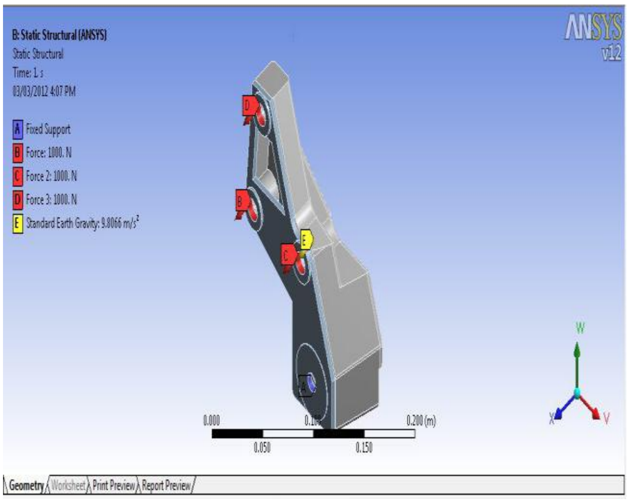

Figure 11: FE Model of Mounted engine bracket.**[1]**

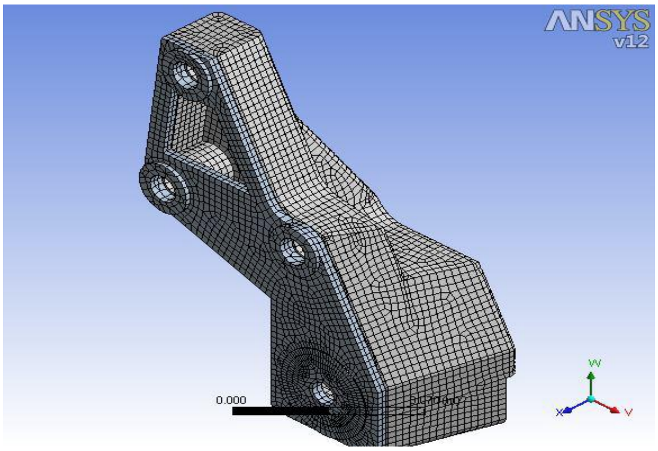

Figure 12: FE Mesh Model of Mounted engine bracket.**[1]**

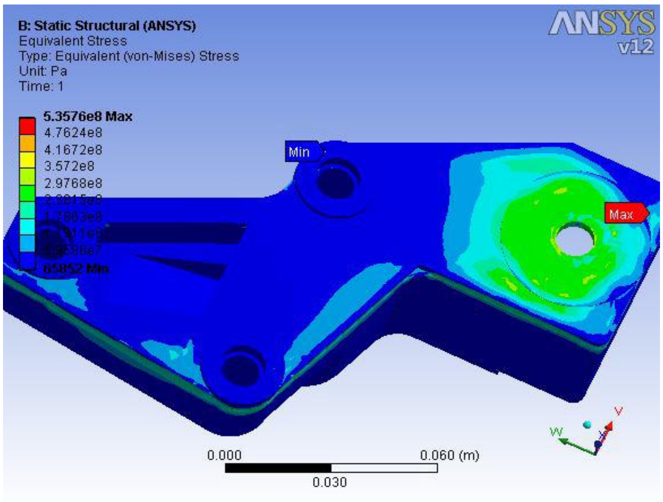

Figure 13: Equivalent (von misses) stresses of 8mm Mesh size[1]
Aluminum Alloy Mounted engine bracket.

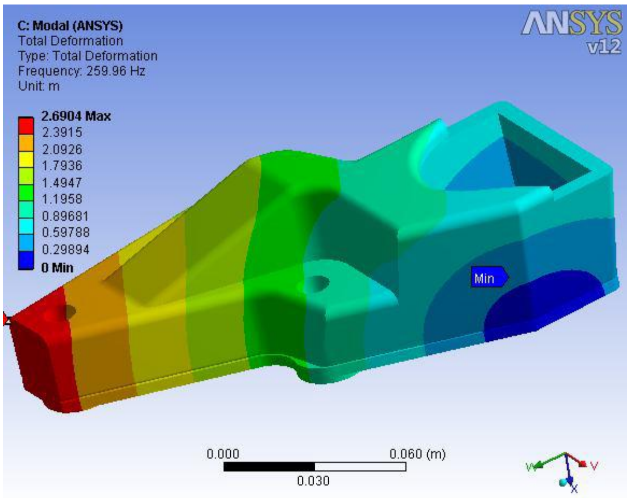

Figure 14: Natural Frequency of 8mm Mesh size Aluminum Alloy Mounted engine bracket.[1]

Table 1: Result Table [1]

Sr .no	Material	Max.Deformation (M)	*Max. Stress (MPa)*	Natural Frequency *(Hz)*
1	AL ALLOY	0.011611	535.76	259.96
2	MG ALLOY	0.027192	366.58	257.95
3	GREY CI	0.0032237	1495.2	198.75

Grey cast iron is the brittle material which reflects the result that the low natural frequency affects the vibration characteristics of the bracket. From the analysis result, AL alloy and MG alloy are the best preferred than the Grey CI, as the natural frequency of the both is nearly same. The effect of the damper is not considered in this analysis.

8. Summary

1) Paper highlights the various types of the mounting brackets and the different types of parameters considered while designing the brackets. Main focus is on the AC compressor mounting bracket for vehicles.
2) Various steps for the designing of the compressor mounting brackets considering the design parameters for engine mounting are discussed.
3) For the optimized design of the compressor mounting bracket the structural topology optimization is studied. Paper also highlights the steps for the topology optimization processes.
4) To validates the process of the structural topology optimization the case study for the mounting bracket is studied in this paper.
5) Case study validates that the structural topology optimization process can minimize the weight of the component without affecting its properties like strength and stiffness. These all process is done taking into consideration the design space for the mounting bracket.
6) The case study of the FEM analysis shows the FEA methods for analysis and the study of the natural frequency of the various material of the engine mounting bracket.

References

[1] 'Umesh S. Ghorpade, D. S. Chavan, Vinaay Patil & Mahendra Gaikawad' "FINITE ELEMENT ANALYSIS AND NATURAL FREQUENCY OPTIMIZATION OF ENGINE BRACKET", IJMIE Vol-2, Iss-3, 2012.
[2] 'Tarundeep Singh Brar, MR. Daljeet Singh' ,"DYNAMIC ANALYSIS OF COMPRESSOR MOUNTING BRACKET OF AUTOMOBILE AIR CONDITIONING SYSTEM", Journal of CAD/CAM & ROBOTICS Vol-2, Iss-3, 2013.
[3] 'G. Phani Sowjanaya, P. Divakara Rao and Dr. C.Udaya Kiran' , "FINITE ELEMENT ANALYSIS OF VIBRATION FIXTURE MADE OF ALUMINIUM AND MAGNISIUM ALLOYS", International Journal of Latest Trends in Engineering and Technology (IJLTET), Vol. 2, 84 – 89, 2013.
[4] 'Pavan B. Chaudhari and Dr. D. R. Panchagade ' , "COMPARISON OF MAGNISIGM, ALUMINIUM AND CAST IRON TO OBTAIN OPTIMUM FREQUENCY FOR ENGINE BRACKET USING FEM", International Journal of Engineering Research and Applications (IJERA) , Vol. 2, 1016-1020. 2012
[5] 'Jeong Woo Chang and Young Shin Lee', "TOPOLOGY OPTIMIZATION OF COMPRESSOR BRACKET", Journal of Mechanical Science and Technology, Vol. 22, 1668-1676. 2008.
[6] Deshpande C., Ghatekar A., "DESIGN OF COMPRESSOR MOUNTING BRACKET USING OPTIMIZATION", International Journal of Advanced Information Science and Technology (IJAIST) Vol.13, No.13, May 2013.
[7] 'Erke Wang, Thomas Nelson and Rainer Rauch' , "A COMPARISON OF ALL-HEXAHEDRAL AND ALL TETRAHEDRAL FINITE ELEMENT MESHES FOR ELASTIC ANALYSIS", Proceedings 4th of international conference, 179-181. 1995