

Dynamic Response Analysis of Compressor Mounting Bracket of an Automobile Vehicle for Aluminum 6061T6

Rajath J K¹ Dr. L Chandrasagar²

¹ Department of Mechanical Engineering, Dr. Ambedkar Institute of Technology Bangalore, India

² Professor Department of Mechanical Engineering, Dr. Ambedkar Institute of Technology Bangalore, India

Abstract—The compressor plays a vital role in the air conditioning system of an automobile. The compressor mounting bracket is a rigid structure which is used to mount the compressor to the engine. Design includes modelling of the bracket by considering all the constraints. Analysis comprises of normal modes analysis and frequency response analysis for aluminium 6061T6. The aim of this work is to find the natural frequency of the bracket for the self-weight by modal analysis and the stresses induced in the bracket due to external excitation by frequency response analysis using FEA method. The analysis is performed using ABAQUS tool and the results are interpreted.

Keywords— Compressor mounting bracket; dynamic analysis; Catia V5; ABAQUS.

INTRODUCTION

In an automobile the air conditioning system consists of a compressor which holds a significant role in air cooling process. The air compressor is connected to the engine by a compressor mounting bracket with nuts and bolts. The main purpose of the bracket is to absorb the vibrations that occur in the system. Hence the bracket is exposed to those parts which produce more vibrations among the different parts of the air conditioning system. To design a vehicle with greater stiffness and simultaneously attain weight reduction is very difficult. The mounting bracket in an automobile used to mount the air compressor to the engine is called automobile compressor mounting bracket. The complications under which the mounting bracket falls is design space issue, material of the product used, performance affected by the density of the product etc.

The compressor is mounted to the engine by a bracket called the compressor mounting bracket. Engine in an automobile is the most vital component of any automobile. Engine is the one which produces heavy vibrations among all the other parts; even the compressor produces more vibrations. The vibrations in an automobile mainly occurs due to the force exerted due to combustion, whirling of flywheel, side forces in piston, reaction forces in the main bearing, opening and closing of inlet and outlet valves, forces between the gears in transmission, forces in belt/chain drives. The mounting bracket should be designed in such a way that it can withstand or take in all the vibrations produced without undergoing failure. When the

word vibration arises the next thing that comes to account is the natural frequency. Each and every body oscillates at a certain frequency called natural frequency, when the working frequency and the natural frequency coincide and it leads to a phenomenon called resonance. Due to resonance, high displacements occur and the structure may fail. Hence it's important to determine the natural frequency of the respective component.

As we know that the natural frequency depends on stiffness of the material and the mass of the respective material. Stiffness is related to young's modulus and mass is related to density. Young's modulus and density are the only inputs used in this analysis. Dynamic response analysis is done by different methods they are; normal modal analysis and modal frequency response analysis. Normal modal analysis is performed to find the natural frequencies and the different mode shapes of the component. Modal frequency response analysis is done by considering an excitation frequency ω which will act as a force, when this ω come to be equal to natural frequency high vibrations occurs because of resonance. In this type of analysis we find the stress in the material acting in all 3 directions.

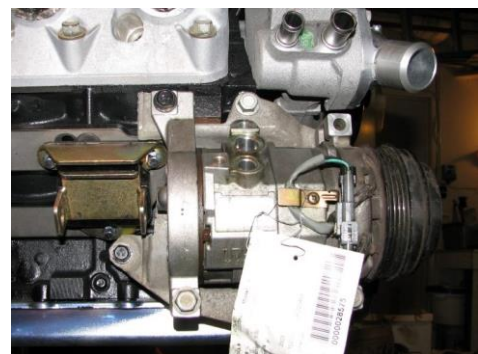


Figure 1.1: Compressor mounted on engine via a mounting bracket

Material

Aluminum 6061T6 is the selected material for the design of mounting bracket. Aluminum is mainly used for its low density and ability to resist corrosion.

Problem Definition

To perform a dynamic response analysis on a compressor mounting bracket in an automobile for aluminum 6061-T6 using the FEA tool ANSYS 15.0 and to find the performance of the bracket for selected material. Optimization of the component is done in order to reduce the weight and improve the performance of the bracket.

Methodology

The first step is to create a 3D geometric model of a mounting bracket using CATIA V5 software. After the geometry is created as per required specifications, the model is then meshed in meshing tool HYPERMESH. The model is discretized into finite number of elements like 2D quad contact elements and the material properties of the model required is selected. The model is constrained by applying certain boundary conditions. The constrained model is then imported into ABAQUS tool in which the dynamic response analysis of the bracket for different materials is performed. The results obtained from ABAQUS for different materials are compared.

Following are the steps involved in Methodology:

- Step 1:** The 3D model of compressor mounting bracket is done using Catia V5, created meshing and analysis has been done using ABAQUS.
- Step 2:** The model is then meshed and boundary conditions are applied using Hypermesh tool.
- Step 3:** Dynamic response analysis is performed using ABAQUS.

Geometric Modeling

Reverse engineering has become a viable technique to make a 3D virtual model by reading 2D drawing. Using Computer Aided Three-dimensional Interactive Application (CATIA) a 3D model of the bracket is created.

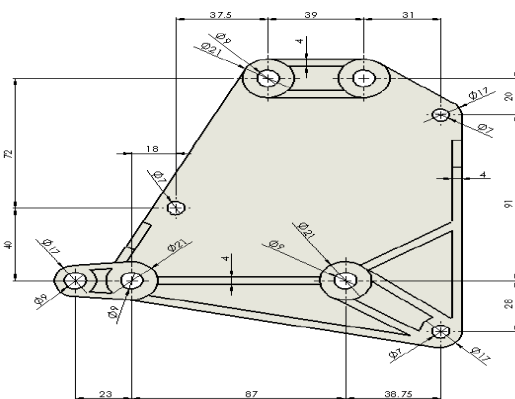


Figure 4.1: Front view of Bracket

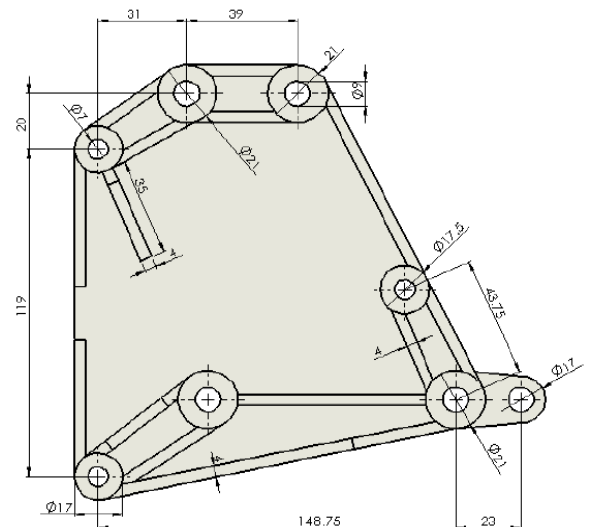


Figure 4.2: Rear view of Bracket

Meshing

Figure 4.3 shows the Finite Element model generated for the geometric model of the mounting bracket using Tetrahedral Element Type (B31 C3D10). The table 4.1 shows the count of number of nodes and elements.

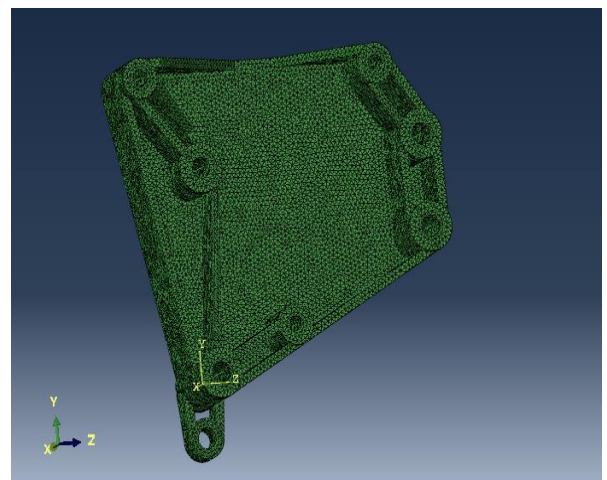


Figure 4.3: Meshed model of the bracket

Table 4.1: Elements and nodes count

Element Types	Tetrahedral Element Type (B31 C3D10)
Number Of Elements	185113
Number Of Nodes	289734

Loads and Boundary Conditions

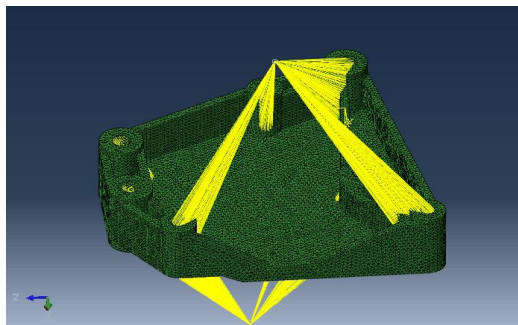


Figure 4.4: Application of loads and boundary condition

Figure 4.4 shows the meshed model of the bracket with applied boundary conditions. Table 4.2 shows the different input values for normal modal analysis.

Table 4.2: Material properties

Material	Young's modulus	Poisson Ratio	Density
Al-6061T6 (aluminum 6061-T6)	66 GPa	0.25	2700 kg/m ³

Following conditions are applied for the normal modes analysis:

- No external force: As per the requirement of natural frequency external forces is not applied.
- Constraints: Natural frequency analysis is carried out as per actual constraint.
- Damping: It is neglected for the natural frequency calculations.
- Output from analysis: The results obtained are magnitude of the frequency and the mode shapes.
- Rigid body element (RBE): Rigid body element can be used on bolt locations to make a rigid region. It does not require a solid bolt, it uses a line element and RBE Elements to represent the stud and bolt respectively, as shown in Fig. 3.10. Tensile, Bending Loads transferred through rigid elements.
- Master node: A node which represents the engine fixture is defines is a "master node" as shown in fig 4.4. This node is an Independent node and was created at the engine mounting position and the mass of the compresses is defines at their center of gravity positions and the loads are transmitted by the rigid body element.

Results & Discussion

The FE model of mounting bracket with applied loads and boundary conditions was analyzed using ABAQUS tool. The

following figure shows the different mode shapes and natural frequencies from normal modal analysis and from the modal frequency response analysis we get the stresses induced in the bracket.

Normal modes analysis

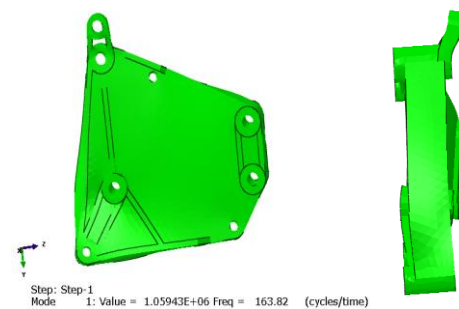


Figure 5.1: Mode 1 of Aluminum (Al-6061T6)

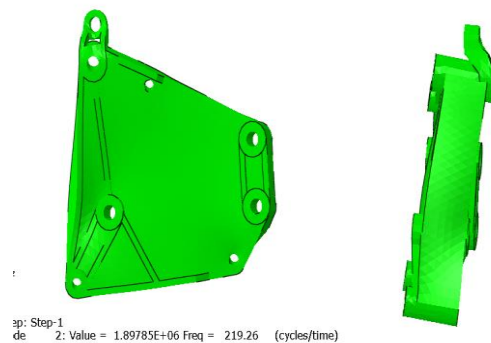


Figure 5.2: Mode 2 of Aluminum (Al-6061T6)

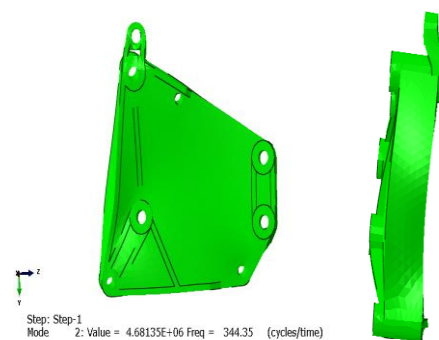
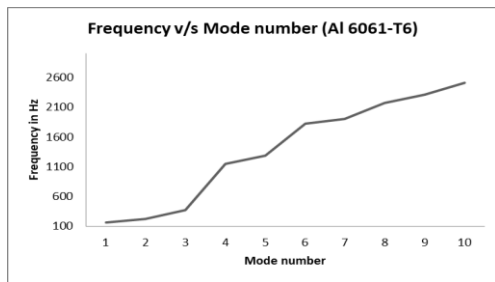


Figure 5.3: Mode 3 of Aluminum (Al-6061T6)

The frequency obtained from the analysis is plotted as shown in the following graphs. The figures show a plot of frequency verses different mode numbers obtained. After determining the natural frequencies of the bracket we proceed for further analysis (Modal frequency response analysis).



Graph 5.1: The plot of frequency vs different modes for AL 6061-T6 material

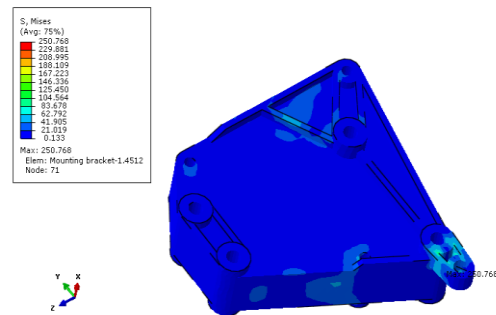


Figure 5.5: Von Mises Stress in mounting bracket along y-axis

Modal frequency response analysis

In the modal frequency response analysis an excitation is given externally in the form of an external force. This force is in the form of $\sin\omega t$, where ω is the frequency of the Excitation. When ' ω ' and the natural frequency of the system become equal, high value vibrations are produced due to resonance. Modal frequency response analysis of compressor mounting Bracket is done along all the three axes. In this analysis, force is applied only in all 3 directions.

- **Boundary Conditions:** For the components of the compressor mounting bracket, material and Property is assigned in the same way as discussed in the previous section. The external excitation is applied in all 3 directions one after the other.
- **Damping Ratio:** The ratio of actual damping coefficient to the critical damping coefficient is known as damping ratio. It is a dimensionless measure describing how Oscillations in a system decay after a disturbance. Damping ratio is taken as 5% for this analysis.

Initial excitation frequency = 161Hz
 Increment for the next frequency =1
 Number of frequencies = 10

The following figures show the stresses acting in the bracket when load applied in each case i.e. x, y and z axis.

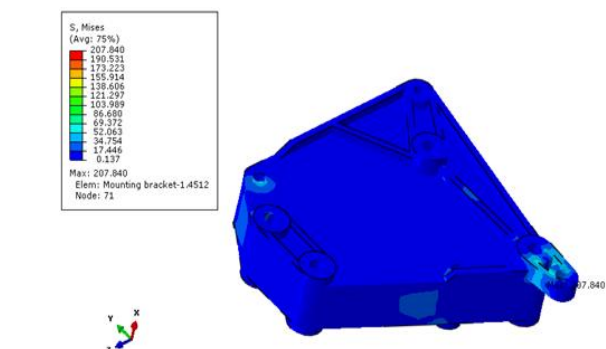


Figure 5.4: Von Mises Stress in mounting bracket along x-axis

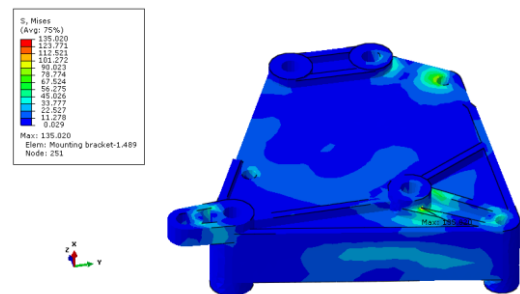


Figure 5.6: Von Mises Stress in mounting bracket along z-axis

Due to dynamic loading with excitation frequency the stresses produced are as low as compared with the yield strength of the material (276 MPa).

CONCLUSION

The vibration analysis is carried on the automobile AC compressor mounting bracket with standard testing condition. The natural frequency mainly depends on the stiffness of structure.

- The first natural frequency of AL6061-T2 is 164 Hz. The resonance of structure is can be avoided if the operating frequency range is between 0 to 163 Hz.
- The model dynamic analysis is carried on AL6061-T2 mounting bracket. The stress developed in structure in all principle direction is lower than the yield strength of material. This show even under resonance the structure is safe under first mode of vibration.

Thus, the use of CAE tools leads to an easy visualization and comparison of data thereby helping in the detection of problems early in the design cycle, reduced number of physical Prototypes resulting in significant saving of time and cost.

REFERENCES

- [1] Topology optimization of compressor bracket, Jeong Woo Chang and Young Shin Lee. Journal of Mechanical Science and Technology April 25, 2008
- [2] Dynamic Analysis of Air Conditioner Compressor Mounting Bracket Vyankatesh D. Pawade, Pushkaraj D. Sonawane. International Engineering Research Journal (IERJ)
- [3] Analysis Of Vehicle Air Compressor Mounting Bracket Murtaza Goawala, Rahul Giri, Niket Phalke, Krishna Singh, Prof. Nitin Sall. International Journal of Recent Trends in Engineering & Research (IJRTER)
- [4]. Analysis and Optimization of Engine Mounting Bracket, Uppe Tejesh and Mrs.T.Nina. International Journal and Magazine of Engineering, technology, Management and Research. Volume:4, Issue:2. Feb-2017.
- [5]. Modal Analysis of Engine Mounting Bracket Using FEA., Sahil Naghate and Sandeep Patil. International Journal of Engineering Research and Applications Vol. 2, Issue4, July-August 2012
- [6]. Comparison of Magnesium, Aluminium and Cast Iron to obtain Optimum Frequency for Engine Bracket using Finite Element Analysis., Pavan B. Chaudhari, Dr. D. R. Panchagade. International Journal of Engineering Research and Applications. Vol. 2, Issue 5, September- October 2012.
- [7]. Finite Element Analysis of Engine Mounting Bracket By Considering Pretension Effect And Service Load., Sandeep Maski, Yadavalli Basavaraj. International Journal of Research in Engineering and Technology. Volume: 04 Issue: 08 August-2015.
- [8]. Finite Element Analysis And Natural Frequency Optimization Of Engine Bracket, Umesh S. Ghorpade, D. S. Chavan, Vinaay Patil & Mahendra Gaikwad. International Journal of Mechanical and Industrial Engineering (IJMIE) ISSN No. 2231 -6477, Vol-2, Iss-3, 2012
- [9]. FEA of Engine Mounting Bracket N.S. Vele, N.K.Kamble, S. S. Sarnobat
- [10]. Design Optimization of Automotive Engine Mount System Dr.Yadavalli Basavaraj, Manjunatha.T.H. International Journal of Engineering Science Invention ISSN (Online): 2319 - 6734, ISSN (Print): 2319 - 6726 www.ijesi.org Volume 2 Issue 3 || March. 2013 || PP.48-53.
- [11]. Analysis and Optimization of Engine Mounting Bracket, Monali Deshmukh, Prof. K R Sontakke. International Journal of Scientific Engineering and Research (IJSER), Volume 3 Issue 5, May 2015