Impact Factor (SJIF): 5.301

International Journal of Advance Research in Engineering, Science & Technology

e-ISSN: 2393-9877, p-ISSN: 2394-2444

Volume 5, Issue 7, July-2018

Design Analysis of Mounting Bracket of Compressor in Automobile Air Conditioning System

Mr. Nitin B. Sall[1], Prof. Tushar Devidas Garse[2]

[1]*Mechanical Engineering, J.T. Mahajan College of Engineering, Faizpur*
[2] *Mechanical Engineering, J.T. Mahajan College of Engineering, Faizpur*

Abstract —***With the automotive air-conditioning industry aiming at higher levels of quality, cost effectiveness and a short time to market, the need for simulation is at an all-time high. In the present work, the use of dynamics analysis technique is proposed in the simulation of the automobile compressor mounting bracket for the vibration loads. The mounting bracket has been analysed using the standard testing conditions. The results revealed that compressor mounting bracket may fail due to resonance in dynamic analysis, but in the static analysis, resonance cannot be predicted under the same magnitude of load. Therefore, dynamic analysis gives a realistic method for its design validation. With the use of the methodology, a new compressor mounting bracket is analysed and optimized. The design analysis would be carried out by using FFT Analyser and Finite Element Analysis software like ANSYS.***

Keywords-compressor mounting bracket; dynamic analysis; natural frequency; mode shapes; FFT analyser; ANSYS

I. INTRODUCTION

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. To reduce these vibratory forces, the compressor is supported by the engine mounting called compressor mounting bracket. From the literature review it is found that a lot of work has been done in the area of vibration analysis for the different parts of an automobile. But, most of the work has mainly focused on **modal analysis**, to find the natural frequency of the component under study and static analysis to evaluate the stresses are as follows:

A. From the studied literature only modal analysis will be done.
B. Frequency response analysis will be done to find the displacements and stresses in the brackets, which is required in a dynamic analysis using FEA.
C. Design Analysis would be carried out by Finite Element Analysis (ANSYS).

II. METHODOLOGY

The analysis for the compressor mounting bracket will be done as follows:

A. CAD modelling of the bracket, compressor and fixture in CAD software.

B. Pre-processing in a CAE software

- Discretization of the geometry using solid elements.
- Application of the boundary conditions (taken from standard testing conditions in an Automotive industry, Standard Engineering PVT. Ltd., Palghar).

C. Solution for normal modes and frequency response analysis.

D. Post-processing – viewing displacements, stresses and interpreting the results.

E. For optimization, reduce the mass from mounting bracket where stresses are negligible.
F. Validation of results by FFT analyser

III. PROPOSED EXPERIMENTAL SETUP

On account of this project, the way and methods to Static analysis and Modal analysis and Dynamic analysis of a mounting bracket is to be studied in Standard Engineering Pvt. Ltd (Palghar).The prospective results of experimental work are aimed to irrecusably investigate the induced stress and fatigue failure of the compressor mounting bracket. A prototype available in the company mentioned would be analysed by using strain gauges, sensors and fatigue test machine which would be then compared with the finite element analysis software ANSYS and with experimental setup used to find the natural frequencies.

Experimental finding of natural frequency and normal mode shape gives the behavior of mounting bracket for free vibrations. Natural frequency and mode shapes i.e. vibrating shapes of bracket at respective natural frequency is obtained by FFT analyzer. There is total six bolt connection available on bracket, which are used for holding the bracket. Modal analysis using FFT analyzer gives natural frequency and amplitude at the same.

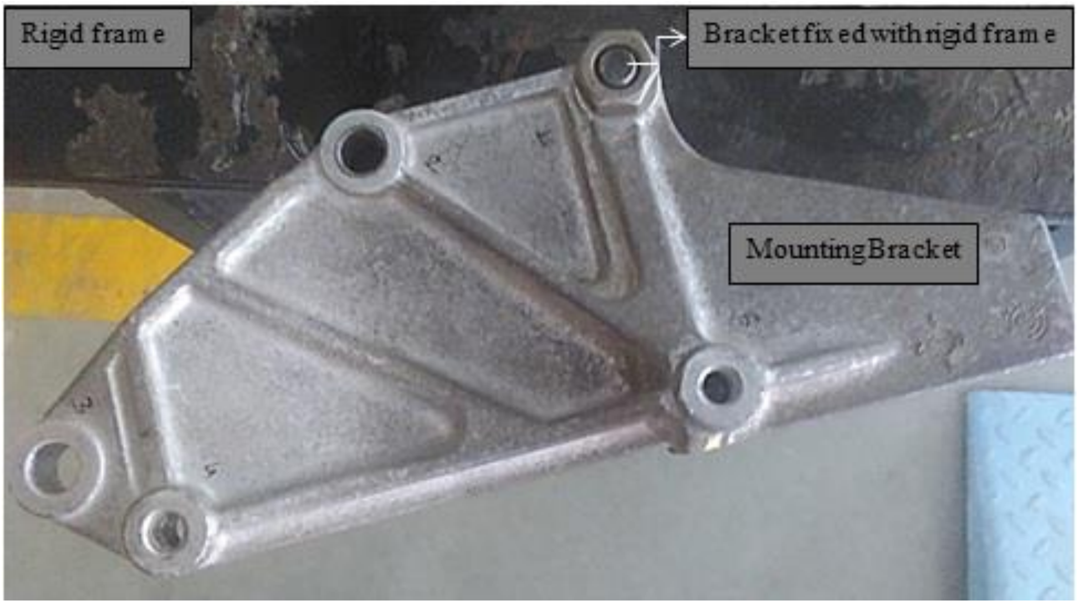

Figure V.1: Compressor Mounting Bracket

Compressor mounting bracket for FFT analysis is retained with rigid frame at different locations as shown in figure 3.1.

Figure V.2: bracket for modal analysis

Vibrational input of free vibrating mounting bracket is taken by transducer i.e. Accelerometer as shown in figure. For different mode shapes and natural frequency determination, bracket is fixed at different bolt locations and FFT output is taken by small disturbance to bracket so as to develop free body vibrations.

IV. ANALYSIS OF COMPRESSOR MOUNTING BRACKET

4.1 Simulation of Compressor Mounting Bracket

This section deals with methodology for simulation and dynamic analysis of compressor mounting bracket. At initial stage there being requirement of three dimensional CAD model with assisting CATIA software and saved in STEP IGES files for FEA analysis. However for CAD model correct dimensions of compressor mounting bracket was opted. Prepossessing work, meshing is done and elements quality checks are observed. Dynamic analysis is done with normal analysis and modal response to observe their behaviour at natural frequencies.

4.2 CAD Model

The Compressor Mounting Bracket component is modelled in 3D modelling software. The Generated CAD model is shown in figure below. CAD model generated is saved in STEP and IGES file extension so as to import in analysis software's. Following is the model and draft figures of compressor mounting bracket.

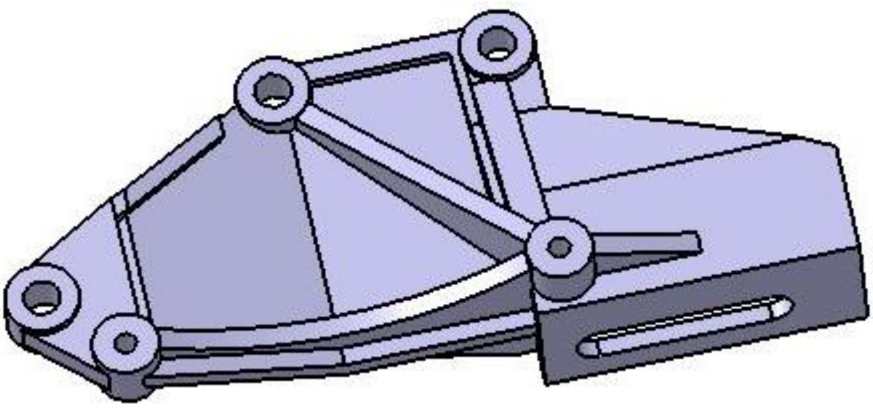

Figure 4.1: 3D CAD model of Compressor Mounting Bracket

4.3 Mesh Generation

Meshing is process of discretising CAD model into fine elements so as to apply boundary conditions and analyse. CAD model in IGES extension is imported for meshing into tetrahedral elements. Meshing of the compressor mounting bracket is done according to the geometry. Mesh quality decides the FEA analysis better performance, so following are the quality checks done for better meshing.

4. 4 Dynamic Analysis

Dynamic analysis for the compressor mounting bracket is done which is divided in two steps:

A. Normal modes analysis.
B. Modal frequency response analysis.

These two steps are related to each other. Output parameters of the first are used as input Parameters of the second step.

4.5 Boundary conditions

The boundary conditions are applied to the meshed model. The different steps are given below.

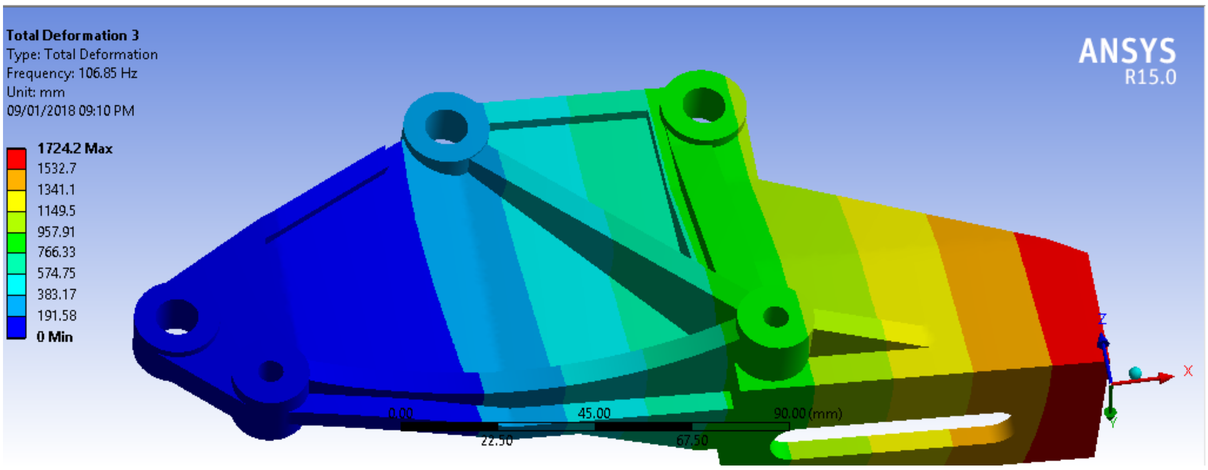

Figure V.2: Mode Shape 01, Frequency 106.85 Hz

Table 4.1: Analysis mode shape frequency

Mode	Frequency (Hz)
1	106.85
2	286.2
3	93.497
4	305.95
5	490.74
6	883.46
7	497.42
8	1028.8

V. RESULT & DISCUSSION

To define a specific material, the material Collectors are used in the software. The material used is "aluminium" with the following properties: linear elastic, isotropic and temperature independent. The values of different properties are:-

A. Modulus of elasticity, E = 6.6 e +04, (N/mm2)
B. Modulus of rigidity, G = 2.7 e +04, (N/mm2)

VI. CONCLUSION AND SCOPE FOR FUTURE WORK

Dynamic analysis gives the realistic and effective results which give the actual effect of performance of any working system. Dynamic analysis - modal analysis is done to get the free vibrational response, determines natural frequency response of the automobile compressor mounting bracket.The results obtained revealed that the high values of stresses deformations are revealed at natural frequencies obtained by modal analysis.

The ribs are one of the most important feature in the bracket design, and can be used in new design effectively to increase the natural frequency and reduce the stress in the bracket, without increasing the mass significantly.
It is also seen that the stresses increase as excitation frequency matches with the natural frequency for the same magnitude of applied load.

Dynamic analysis using CAE tools will save time money and other resources so as to detect vibrational, responses, Deformation and stress response at respective frequency will give the modal sustenance at respective national frequency.Experimental investigation leads to cast and not possible to have more iterative. While this limitations are not with CAE tools. CAE tools can be used to have number of iteration which lead to go "n" number of iterations that a researcher intend top di.

6.1 Scope For Further Work

The present work can be extended in the following ways:

A. Other automotive parts can be analysed, and can be optimized by this methods.
B. Other CAE software's can be used for optimization of different automotive parts.
C. The methodology can be practically validated for the value of stresses by using strain gauges, on an electro-dynamic shaker.
D Thermal analysis can also be benefitted to get the better optimization of mounting brackets with dynamic analysis.

REFERENCES

[1] G. Phani Sowjanya, P. Divakara Rao and Dr. C. Udaya Kiran, "Finite Element Analysis of Vibration Fixture Made of Aluminum and Magnesium Alloys",International Journal of Latest Trends in Engineering and Technology (IJLTET),Vol. 2, pp.84 – 89,(2013)

[2] Sahil Naghate and Sandeep Patil, "Modal Analysis of Engine Mounting Bracket Using FEA", International Journal of Engineering Research and Applications (IJERA), Vol. 2, pp.1973-1979,(2012).

[3] Pavan B. Chaudhari*, Dr.D. R. Panchagade," 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) ISSN: 2248-9622 Vol. 2, Issue 5, September- October 2012, pp.1016-1020

[4] Jeong Kim, Joo-Cheol Yoon and Beom-Soo Kang (2007), "Finite element analysis and modeling of structure with bolted joints". Applied Mathematical Modelling, Vol. 31, pp. 895-911.

[5] Doo-Ho Lee, Jeong-Woo Chang and Chan-Mook Kim (2003), "Optimal Shape Design of an Air-Conditioner Compressor Mounting Bracket in a Passenger Car", Vol 01, pp.1667–1672.

[6] Sheldon Imaoka (2002), “Constant equation - CERIG, RBE3 and RIGID 184”, Proceedings of ANSYS conference, USA.

[7] Erke Wang, Thomas Nelson and Rainer Rauch (1995), “A comparison of all-Hexahedra and all Tetrahedral Finite Element Meshes for elastic analysis”, Proceedings 4th of international conference,pp. 179-181.