Optimization of Compressor Mounting Bracket of car under dynamic conditions

Sachin Kalsi
Chandigarh University,Mohali,
sachin.uie@cumail.in

Abstract :
With the expansion of rivalry in the field of car ventilating enterprises, they are searching for quality, cost adequacy, brief time to showcase and simple approach to take care of their issues. In this way, a recreation requires is at an abnormal state. 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.

INTRODUCTION

The design and development of an automobile is a big and complex task for a company or a designer team to fulfil all the requirements of a vehicle according to customer need and different standards adopted by various authorities in different countries 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. The continual development, improvement in quality and incorporation of adding more additional features without mush variation in the market price has proved to very effective in the highly competitive field of automobiles. 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.

A CAE program is a numerical model written in a programming dialect utilizing an arrangement of calculations that characterize the assembling forms. There are numerous commonsense issues in building field for which we can't without much of a stretch acquire correct arrangements. The powerlessness to accomplish a correct arrangement might be ascribed to either the intricate idea of representing respectful conditions or the troubles that emerge from managing the limit and introductory conditions. To manage such issues, we require help of some numerical approximations. Diagnostic arrangements demonstrate the correct conduct of a framework anytime inside the framework yet numerical arrangements of rough correct arrangements just at discrete focuses.

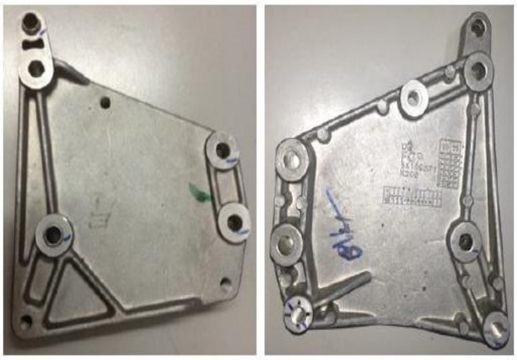
LITERATURE REVIEW

Many researchers have worked in the field of vibration analysis of many components using finite element analysis approach. **Chang and Lee [1]** performed topology dvancement of compressor section. Base section display for the topology streamlining was demonstrated by considering the obstruction with the contiguous vehicle parts.Chaudhari and Panchagade[2] thought about various materials that are magnesium, aluminum and cast iron to acquire advanced characteristic recurrence for motor section utilizing limited component investigation approach. Choi et al. [3] done an investigation on the quickened vibration continuance trial of battery settling section mounted in electrically determined vehicles. **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.

CAD MODEL AND MESHING OF BRACKET

C. 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

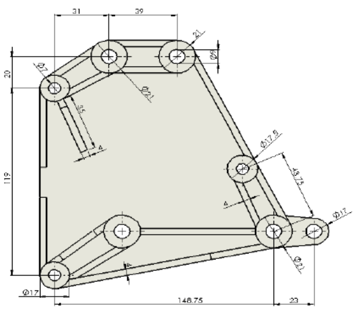
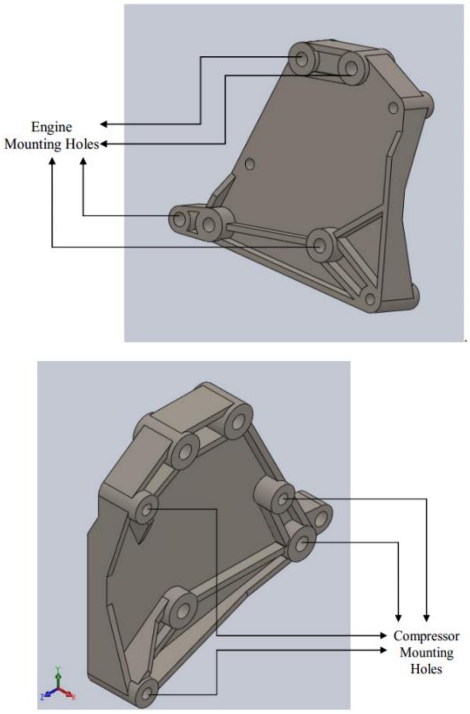


Fig. 2: Dimensions of bracket after reverse engineering

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

D. *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.

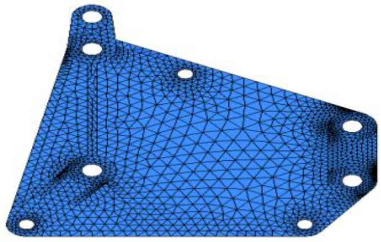


Fig. 4: Tetrahedral meshing of bracket

XII. DYNAMIC ANALYSIS OF BRACKET

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

A. *Normal modes analysis*

This analysis was done to find out the natural frequencies and mode shapes of the bracket. The material chosen for this analysis was aluminium.

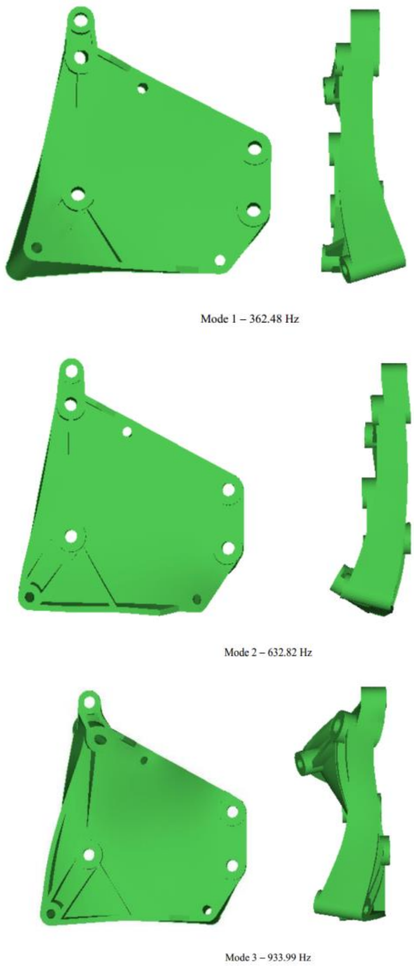


Fig. 4: Different modes of bracket

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

B. Modal frequency response analysis

In the modal frequency response analysis an external harmonic excitation 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.

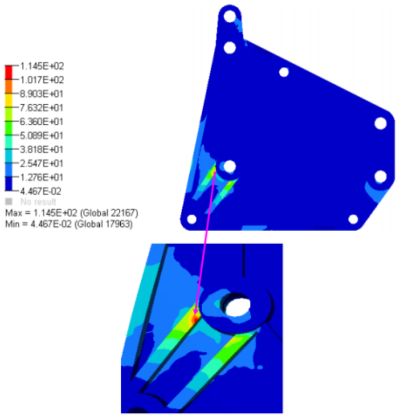


Fig. 5: Von misses stress in X axis

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.

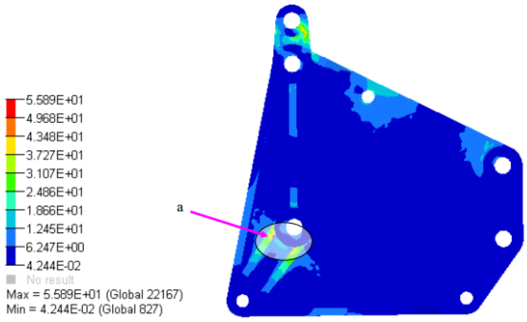


Fig. 6: Von misses stress in Y axis

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.

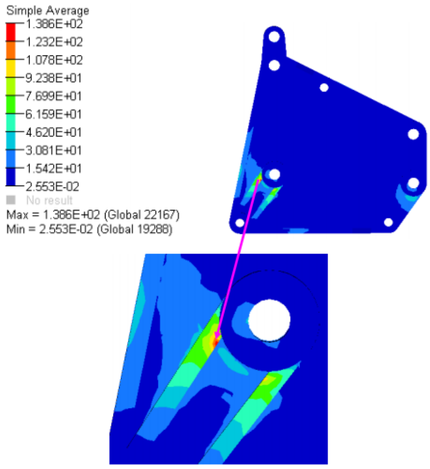


Fig. 7: Von misses stress in Z axis

XIII. 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.

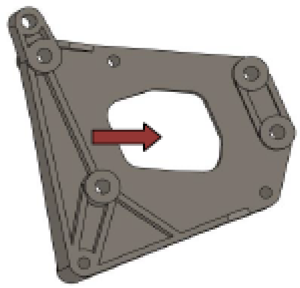


Fig. 8: Modified mounting bracket

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

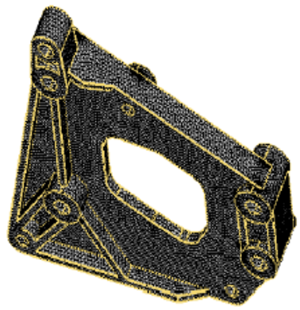


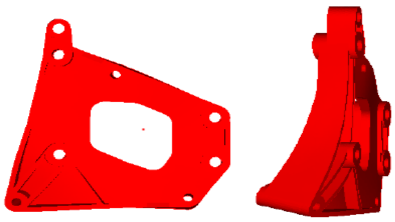
Fig 9: Tetrahedral meshing of the modified mounting bracket

B. DYNAMIC ANALYSIS OF MODIFIED BRAKCET

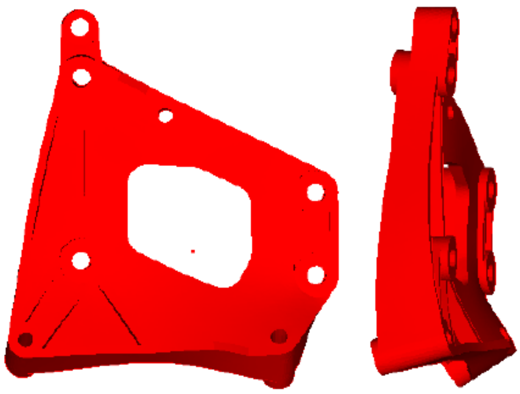
Dynamic analysis for the compressor mounting bracket is done using Normal modes analysis.

Normal Modes Analysis of the Mounting Bracket

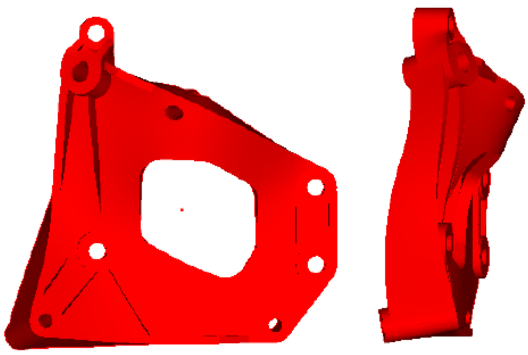
The procedures of normal modes analysis of modified mounting bracket are same as explained earlier. The results obtained are:



Mode 1 – 359.91 Hz



Mode 2 – 625.44 Hz



Mode 3 – 914.41 Hz

Fig 10: Natural frequencies and mode shapes

XIV.CONCLUSION

The use of CAE tools leads to an easy visualization and comparison of data therebyhelping in the detection of problems early in the design cycle, reduced number of physicalprototypes resulting in significant saving of time and cost and last but not the least, moredesign iterations by incorporating simulation techniques. accomplished for sections diminishing the mass of the same by around 9.4%. It is likewise observed that the burdens increment as excitation recurrence matches with the common recurrence for a similar greatness of connected load.

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.