Dynamic Analysis of Compressor Mounting Bracket of Automobile Air Conditioning System

P. Arjun Sagar, G. Sarath Kumar, M V Ramesh

Department of Mechanical Engineering ,Brahmaiah College of Engineering, Nellore, ,INDIA

A B S T R A C T

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 analyzed using the standard testing conditions. The results revealed that the 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.

Keywords: Dynamic Analysis, ANSYS, Compressor Mounting Bracket

1. Introduction

Compressor is the main functional component in the refrigeration cycle. Compressor is assembled on the compressor mounting plate. This subassembly is then assembled on the refrigerator body in the rear bottom side of refrigerator. In the existing design compressor mounting plate is assembled on the refrigerator body using two fastener locations. Compressor being the dynamic component generates vibration and noise. Vibration is transferred to refrigerator body through mounting plate. During the operating cycle, compressor will also exert harmonic pulsating forces on the mounting plate. Frequency of these forces is the operating frequency of compressor (1500 RPM to 4500 RPM).

Fig 1.Compressor

* *Corresponding author.*
E-mail address: r082146@gmail.com

1.1 Introduction to Vibration

Vibration is the motion of a particle or a body or system of connected bodies displaced from a position of equilibrium. Most vibrations are undesirable in machines and structures because they produce increased stresses, energy losses, cause added wear, increase bearing loads, induce fatigue, create passenger discomfort in vehicles, and absorb energy from the system. Rotating machine parts need careful balancing in order to prevent damage from vibrations.

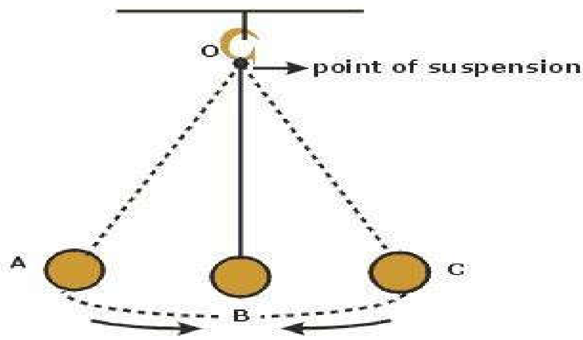


Fig – 2 Swinging of simple pendulum

In this way, vibratory motion is repeated indefinitely and exchange of energy takes place. Thus, any motion which repeats itself after an interval of time is called vibration or oscillation. The swinging of simple pendulum as shown in Fig. 2 is an example of vibration or oscillation as the motion of ball is to and fro from its mean position repeatedly.

The main reasons of vibration are as follows:
a) Unbalanced centrifugal force in the system. This is caused because of non-uniform material distribution in a rotating machine element.
b) Elastic nature of the system.
c) External excitation applied on the system.
d) Winds may cause vibrations of certain systems such as electricity and telephone lines, etc.

1.2 Importance of Vibration study in Engineering

The structures designed to support the high speed engines and turbines are subjected to vibration. Due to faulty design and poor manufacture there is unbalance in the engines which causes excessive stresses in the rotating system because of vibration. The vibration causes rapid wear of machine parts such as bearings and gears.

2. Literature Review

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. A review of the literature related to the design and analysis with a focus on vibration analysis of these mounting brackets is presented here. G. PhaniSowjanyaet al [1] performed a study on vibration parameters to test the avionic equipment. Vibration is the most important mode of failure in avionic equipment. The avionic equipment fitted into the aircraft has to withstand high vibrations.

3. Introduction to CAD/CAM/CAE

The Modern world of design, development, manufacturing so on, in which we have stepped can't be imagined without interference of computer. The usage of computer is such that, they have become an integral part of these fields. In the world market now the competition in not only cost factor but also quality, consistency, availability, packing, stocking, delivery etc. So are the requirements forcing industries to adopt modern technique rather than local forcing the industries to adapt better techniques like CAD / CAM / CAE, etc.

This penetration of technique concern has helped the manufacturers to

a) Increase productivity
b) Shortening the lead-time
c) Minimizing the prototyping expenses

d) Improving Quality
e) Designing better products

What is CATIA - CATIA is mechanical design software. It is a feature-based, parametric solid modeling design tool that takes advantage of the easy-to-learn Windows graphical user interface. You can create fully associative 3 -D solid models with or without constraints while utilizing automatic or user-defined relations to capture design intent.

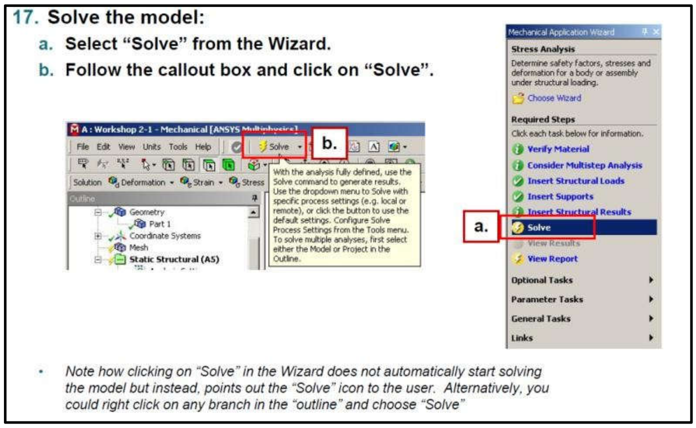


Fig. 3 Ansys Mechanical – Preprocessing-5

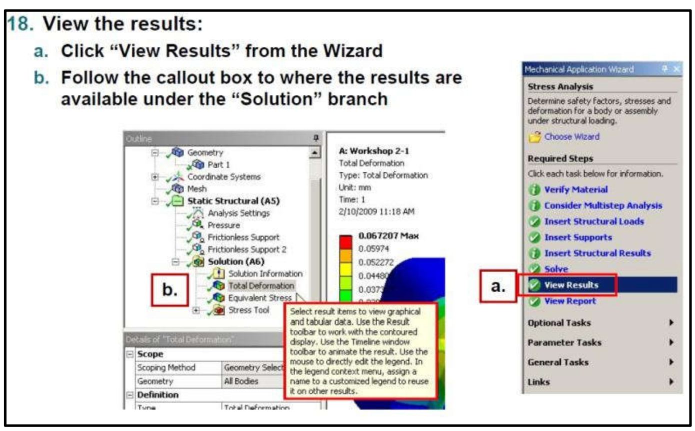


Fig. 4 Ansys Mechanical – Solution

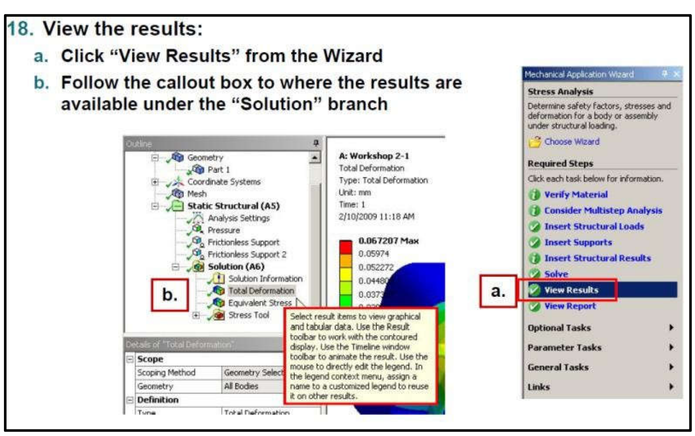


Fig 5 Ansys Mechanical – Results

4. Objective and Problem Statement

Objective

The objectives of the project are as follows

- To develop structural modeling of Mounting Bracketusing CATIA
- To perform finite element analysis of Mounting Bracketby using Ansys 15 for structural steel material
- Investigate the maximum stress of Mounting Bracketusing ANSYS 15 software
- Results description in terms of factor of safety, stiffens, deformation and stress.

Problem Statement

During Gantry Hoist, engineers have to manually evaluate various tools such as CAD and CAE tools. This process takes considerable amount of time and effort. Furthermore, the process of FEM simulations such as meshing and post processing is very iterative and time consuming. In this work, an alternative way to perform FEA will be presented. The main objective of this approach is to relief engineers from time consuming and iterative work.

5. Design and Analysis

5.1 The following objectives are laid in order to achieve the best solutions possible.

1. **Creation of structure.** To be able to analyse in a fast way, different geometries of the same model, it is necessary to be able to modify its dimensi on. However, the mass properties of this model should be similar to the real model in order to obtain similar results when performing a structural analysis.

2. **FE Process.** In order to dimension the structure it is necessary to know the stress distribution, this operation is time consuming and it depends on the complexity of the design. When similar models are evaluated, it is required to create a FE-model for each design, increasing the overall time of the design process. The main objective of this thesis is to automate the Finite Element Analysis.

3. **Creation of a Graphical User Interface (GUI).** A GUI is required to control the different stages of the design process, by integrating design tools. A framework which facilitates the user to modify the different parameters which have an impact in the FE-simulation has to be established.

4. **Methods comparison.** An important objective is the comparison of the automatic methodology with the manual, in order to establish the benefits from the proposed automatic methodology.

5. **Validation**

The results from the FEM simulations performed on the Gantry Hoist CAD model needs to be validated by comparing with material strength. Stresses and deformation are the main criteria for the comparison.

5.2 Simulation of Compressor Mounting Bracket

This section discusses the methodology of analysis of the compressor mounting bracket.

Following steps were performed for the simulation of the bracket:

- CAD model was generated with the help of reverse engineering of the compressor mounting bracket (physical part).
- Mesh was generated for the analysis.
- Dynamic analysis (normal modes and modal frequency response) was performed.

5.3 Reverse Engineering

As computer-aided design (CAD) has become more popular, reverse engineering has become a viable method to create a 3D virtual model of an existing physical part for use in 3D CAD, CAM, CAE or other software. The reverse-engineering process involves measuring an object and then reconstructing it as a 3D model.

Properties of Outline Row 3: Structural Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	7850	kg m^-3		
3	Isotropic Secant Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
7	Derive from	Young's Modu...			
8	Young's Modulus	2E+11	Pa		
9	Poisson's Ratio	0,3			
10	Bulk Modulus	1,6667E+11	Pa		
11	Shear Modulus	7,6923E+10	Pa		
12	Alternating Stress Mean Stress	Tabular			
16	Strain-Life Parameters				
24	Tensile Yield Strength	2,5E+08	Pa		
25	Compressive Yield Strength	2,5E+08	Pa		
26	Tensile Ultimate Strength	4,6E+08	Pa		
27	Compressive Ultimate Strength	0	Pa		

Figure 6 - Material mechanical properties

The following steps are time consuming and iterative and require constant supervision from the user, hence the reason to automate them.

5.2.3. Meshing.

One of the most relevant steps in the Finite Element Analysis is the meshing. The speed and the accuracy of the results have a direct connection in how this part is done. The higher the numbers of nodes are the higher the accuracy of the results, however the speed of the simulation decreases. Figure 7 shows how the mesh looks in ANSYS Mechanical.

Mesh diagram should be placed.

Tetrahedrons second order mesh is used for the structure. Body sizing of 5mm used for the structure.

Total No of Nodes: 62372

Total No of Elements: 41582

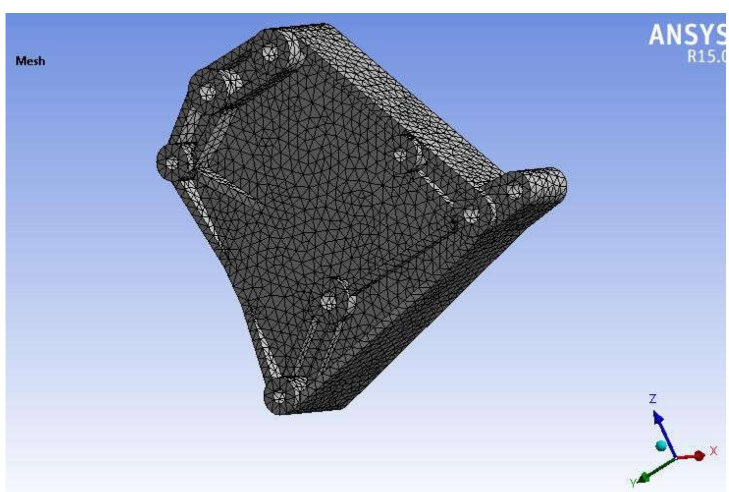


Fig.7 Mesh of the structure

5.2.4. Pre-processing.

After meshing the structure, the Boundary Conditions have to be applied in the model. For obtaining the stress the algorithm first calculates the displacements, hence the necessity to fix the model. Furthermore, after fixing the model the load conditions that influence the structure are given as inputs to the analysis. In Figure 9 it is possible to observe how these boundary conditions are placed in the structure.

5.2.5. Loads

Compressor weight of 100kg considered and is applied as point mass in the structure. Modal analysis will be ran to calculate the frequencies of the structure.

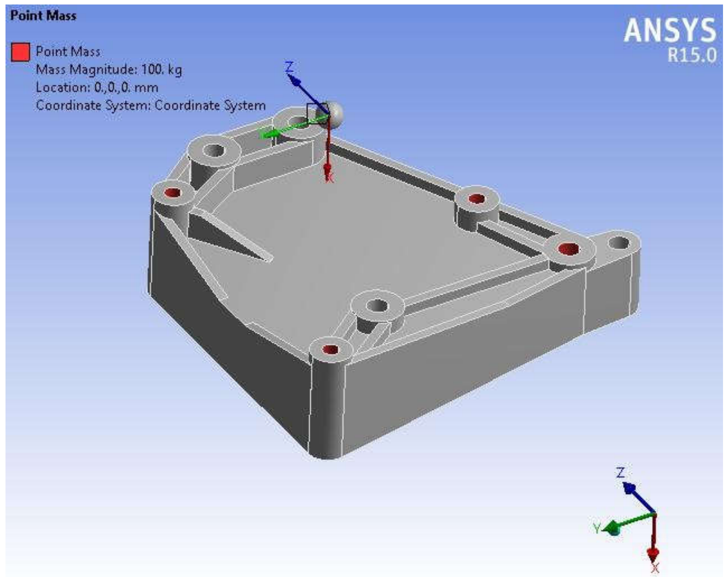


Fig. 8 Load applied in case1

5.2.6. Boundary conditions

Remote displacement with all translations and rotations fixed at the highlighted surfaces in the figure

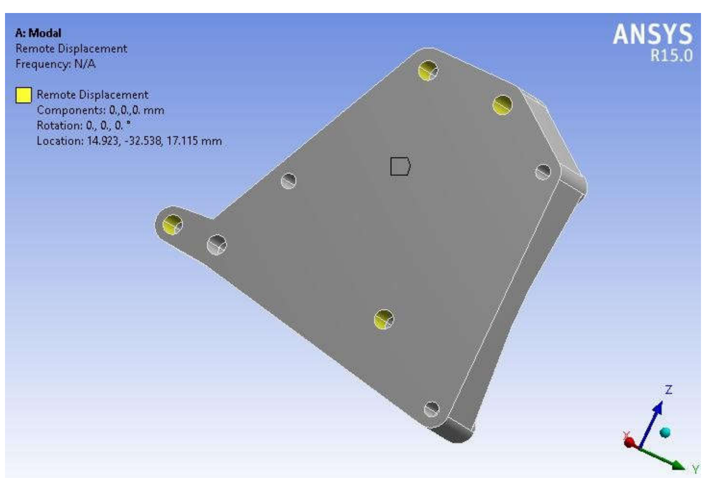


Fig. 9 Boundary condition applied in the structure

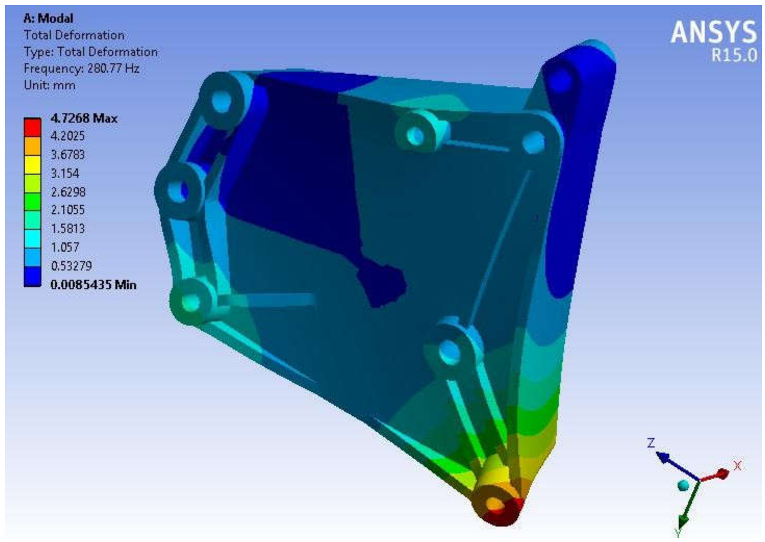
5.2.7. Post processing.

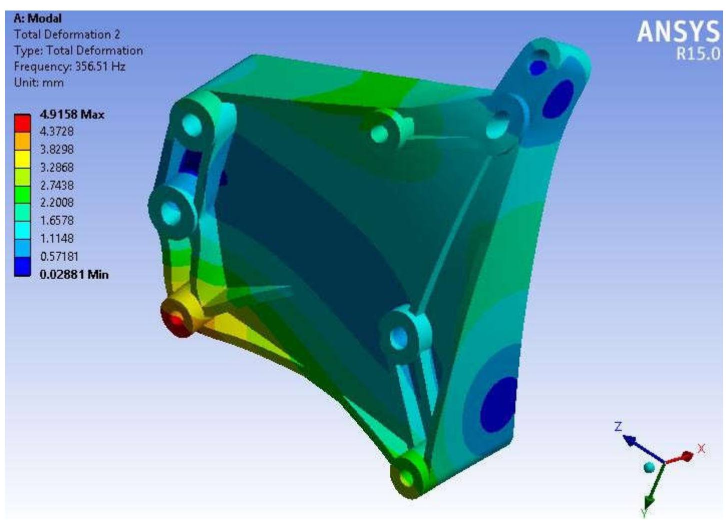
The final step is to run the simulations, but before it has to be specified which results are required by the user. In order to determine if the model can resist the loads applied to it, it is necessary to know, e. g. the Maximum Von Mises stress and the displacement. Knowing these results the user can compare with the data from the material used and applying the safety factor it can be determined if the structure is stiff enough. Another use is being able to extract the results automatically for the possibility to optimize the structure.

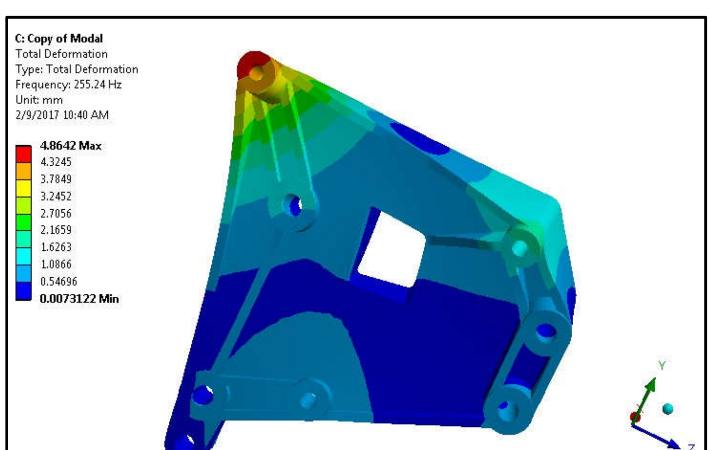
6. Results and Discussions

This section is intended for presenting the results obtained after learning the theories and applying the method described in the two previous chapters. The frequencies for the 8 modes are calculated from the FEA results and is given below

1st Mode:







7. Conclusion

The frequency results of actual geometry and modified geometries are as expected and below the limits. The operating frequency range for t he mounting bracket is 25Hz-75Hz. The first fundamental frequency calculated through dynamic analysisfor the designed mounting bracket is 281 Hz and for optimized model is 255Hz which are above the operating frequency range and hence the design is safe.the material removed from the actual geometry also getting almost same results of actual geometry because of removal of material from the model the manufacturing cost will be r educe without strength effect on the original geometry.

REFERENCES

[1] Nouby M. Ghazaly, Sufyan Mohammed and Ali M. Abd-El-Tawwab (2012), “International Journal of Engineering Research and Applications (IJERA)”, Vol. 2,241 - 250.

[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, 1016-1020.

[3] SahilNaghate and Sandeep Patil (2012), “Modal Analysis of Engine MountingBracket Using FEA”, International Journal of Engineering Research and Applications (IJERA), Vol. 2, 1973-1979.

[4] Jeong Woo Chang and Young Shin Lee (2008), “Topology optimization ofcompressor bracket”, Journal of Mechanical Science and Technology, Vol. 22, 1668-1676.

[5] AsheberTechane (2007), “Dynamics and Vibration analysis of bus body structures”, Proceedings of MSC/NASTRAN conference, Brazil.

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

[7] Doo-Ho Lee, Jeong-Woo Chang and Chan-Mook Kim (2003), “Optimal ShapeDesign of an Air-Conditioner Compressor Mounting Bracket in a Passenger Car”, Vol 01, 1667–1672.

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

[9] 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, 179-181.

[10] Yasuo O., Nagashima H. and Tanahashi, "Vehicle technology handbook tests andevaluation", Society of Automotive engineers of Japan.