ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Analysis and Optimization of Engine Mounting Bracket

Uppe Tejesh
M.Tech (CAD/CAM),
Department of Mechanical Engineering
Malla Reddy College of Engineering
Maisammaguda, Dhulapally, Secunderabad,
Telangana, India.

Mrs.T.Nina
Assistant Professor
Department of Mechanical Engineering
Malla Reddy College of Engineering
Maisammaguda, Dhulapally, Secunderabad,
Telangana, India.

ABSTRACT

The engine mounting plays an important role in reducing the noise, vibrations and harshness for improving vehicle ride comfort. The brackets on the frame that support the engine undergo high static and dynamic stresses as well as huge amount of vibrations. Hence, dissipating the vibrations and keeping the stresses under a pre-determined level of safety should be achieved by careful designing and analysis of the mount brackets. Keeping this in mind in this paper, Static Analysis, model analysis of engine mounting is done as well as harmonic response analysis of bracket as a part of dynamic analysis is performed with the FEA software package ANSYS 15.0.The existing model is optimized and a novel model was proposed to reduce the weight of the rib of the engine mounting bracket as well as the harmonic response in term of acceleration is checked to ensure that the proposed model will not result in to noisy operation. The results of the stresses, deformations and harmonic response for the both models of the engine mounting bracket were compared to each other. With the proposed model of the engine mounting bracket 12.5%weight reduction is achieved maintaining an acceptable level of yield stress and harmonic response

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). If the exciting force matches the natural frequency of the material, resonance occurs. Resonance will be characterized by excessive vibration amplitude and noise. Since refrigerator is a household appliance, any noticeable noise level increase from refrigerator leads to negative impact on perceived quality of the product and hence impacts sales. Further, noticeable noise level increase will also cause the customer to raise a service call and hence the company has to bear additional service cost

Compressor mounting plate is also very close to the floor of the house since it is present is bottom portion of refrigerator. Compressor considered has a weight of 12 Kg. Hence compressor mounting plate has to support this dead weight. Due to the weight, there will be some deformation in the plate. It is very important that this deformation is not excessive since plate will touch the floor in case of excessive deformation. Thus the compressor mounting plate has two main functions: 1. To minimize vibrations and prevent resonance in any case 2. To minimize static deformation

The compressor plays a very important role in the automotive air conditioning system. It is attached to the engine via a compressor bracket and tightened by bolts. The compressor bracket is exposed to the heaviest vibration conditions among the air

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

conditioning parts. Present work deals with FEA analysis of compressor bracket using ANSYS.

Fig-1: Compressor

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.

Vibration occurs when a system is displaced from a position of stable equilibrium. The system tends to return to this equilibrium position under the action of restoring forces. A dynamic system composed of a finite number of storage elements is said to be lumped & discrete, while a system containing elements, which are dense in physical space, is called continuous system. The analytical description of the dynamics of the discrete case is a set of ordinary differential equations, while for the continuous case it is a set of partial differential equations. The analytical formation of a dynamic system depends upon the kinematic or geometric constraints and the physical laws governing the behavior of the system.

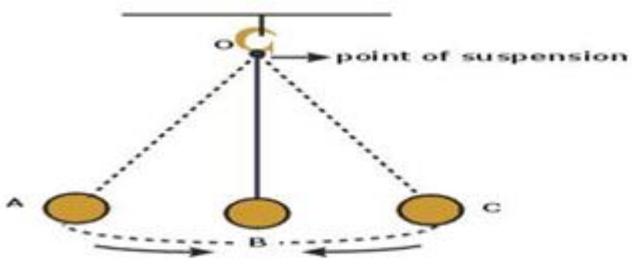


Fig – 2: Swinging of simple pendulum

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:

- Unbalanced centrifugal force in the system. This is caused because of non-uniform material distribution in a rotating machine element.
- Elastic nature of the system.
- External excitation applied on the system.
- Winds may cause vibrations of certain systems such as electricity and telephone lines, etc.

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. Unwanted vibrations may cause loosening of parts from the wheels of locomotive can leave the track due to excessive vibration which results in accident or heavy loss. Many buildings, structures and bridges fall because of vibration.

Vibration can be used for useful purposes such as vibration testing equipments, vibratory conveyors, hoppers, sieves and compactors. Thus undesirable vibrations should be eliminated or reduced upto certain extent by the following methods:

- Removing external excitation, if possible.
- Using shock absorbers.
- Dynamic absorbers.
- Resting the system on proper vibration isolators.

2. DESIGNING THE COMPRESSOR USING CAD AND CATIA

2.1 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.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

- Dynamic analysis (normal modes and modal frequency response) was performed.

2.2 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. The physical object can be measured by using:

- **3D scanning technologies** like Coordinate-measuring machine (CMMs), laser scanners, structured light digitizers, or industrial CT Scanning (computed tomography).

- **Manual measuring devices** such as flat surface plate or table, level meter, scriber, vernier caliper, square, divider, scale, depth gauge etc.

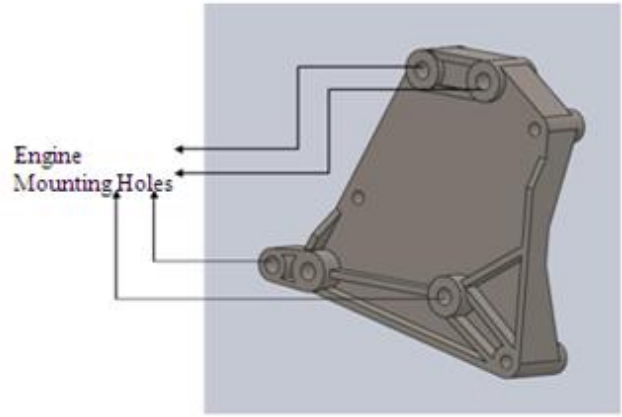
Reverse engineering of the compressor mounting bracket, is done by using manual measuring devices. The detailed drawings are shown in the Fig. 3.

Fig-3: Compressor mounting bracket

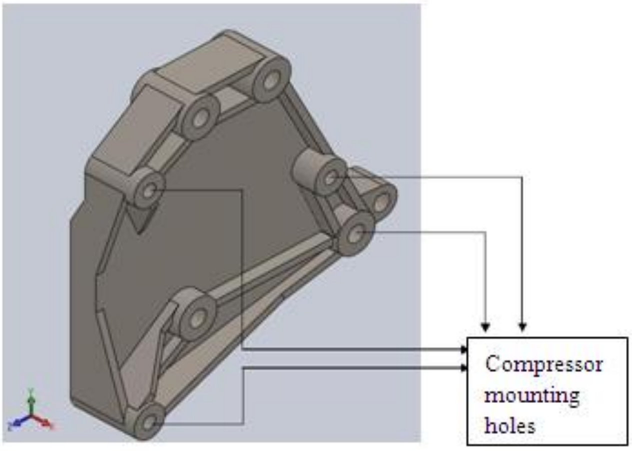
CAD Model

From the reverse engineered data, the component is modeled in Solid Works software. The generated CAD model is shown in Fig. 4.

Solid Works has .SLDPRT extension which cannot be directly read by the CAE software due to data losses. Thus, the file (.SLDPRT) was converted into a neutral file format for the data transfer (like IGES, Para-solid, and Step etc.)



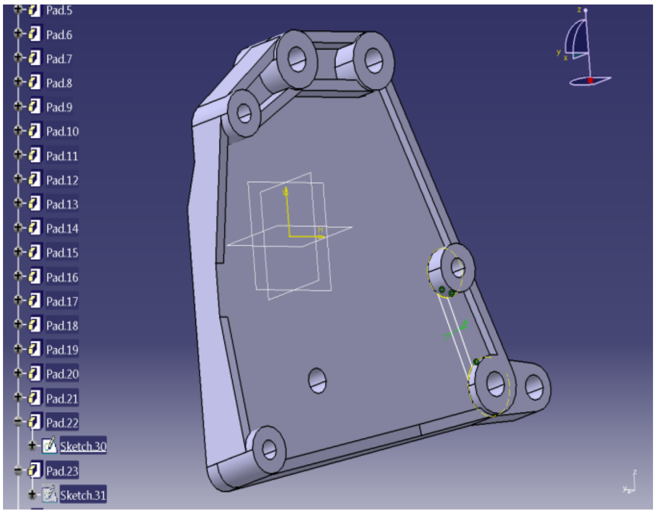
a.) Engine mounting location



b.) Compressor mounting location

Fig-4: CAD model of mounting Bracket

CATIA Model



ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

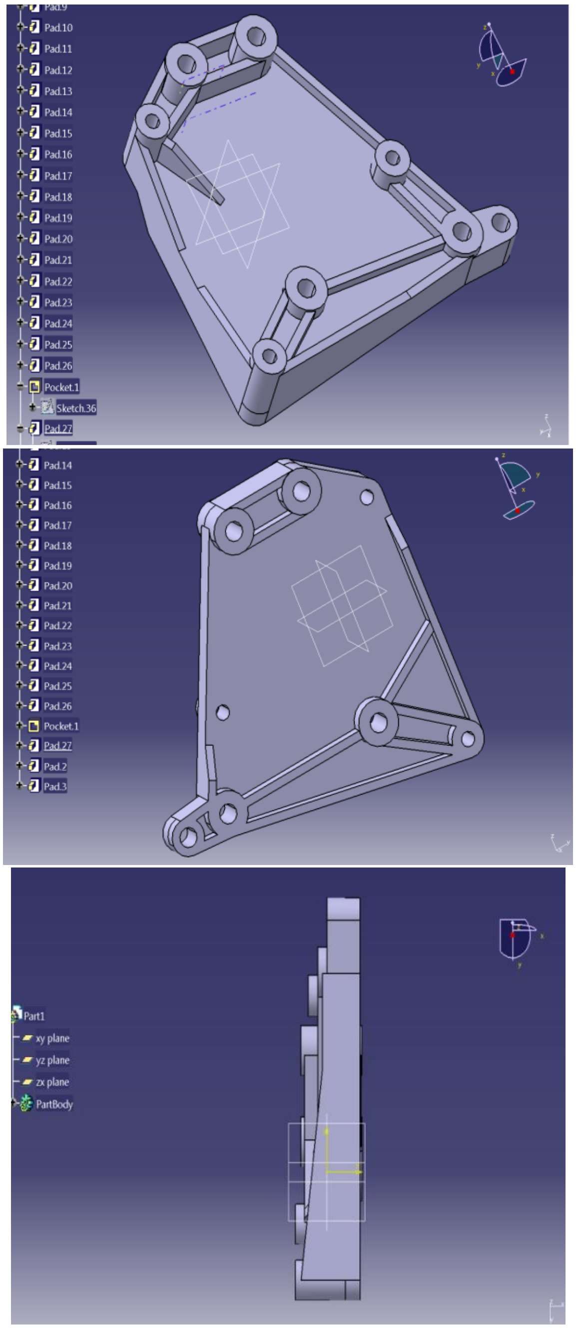


Fig -5: CATIA Modeling

3. PERFORMING STRUCTURAL ANALYSIS IN ANSYS

In this section, the steps taken to perform a structural analysis in ANSYS are explained. It is necessary to identify the tedious and time consuming steps and try to automate them to reduce the FE simulation time and to avoid the constant interaction of the user with the FE tool. Following the list of steps are presented.

Geometry: The first step to take in order to perform the analysis is to define the geometry to be evaluated. This geometry is normally done in CAD software and later imported into a dedicated FE-program.

ANSYS WORKBENCH can be combined with the dedicated FE-tool ANSYS Workbench where it is possible to create the Name Selection; this mean to assign names to surfaces, lines or points from the geometry that can be used later on in the analysis.

Manual importation of CAD models requires the user to browse between different folders or opening ANSYS Workbench and pressing the Workbench button. This procedure requires automation to avoid spending time in the tasks explained above.

Material: After having the geometry defined, the next step is to assign a material to this geometry. Depending on the type of analysis some properties have more importance than others. For a structural analysis the Young's modulus and the Poisson ratio are the most important. The importance of automating this step is to avoid the need of manually selecting the required material from a long list located in ANSYS, especially when the user knows beforehand the name of the material.

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		

Table-1: Material mechanical properties

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

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. Fig-6 shows how the mesh looks in ANSYS Mechanical.

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

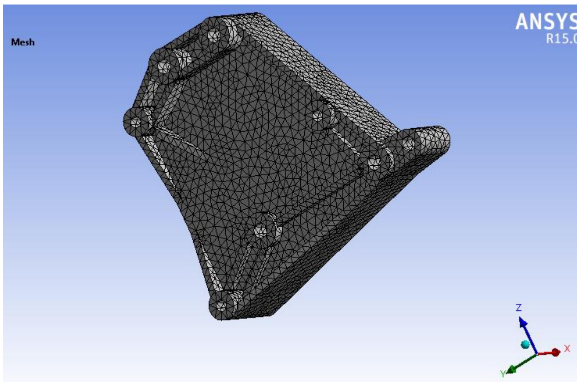


Fig-6: Mesh of the structure

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.

Loads: Compressor weight of 100kg considered and is applied as point mass in the structure.

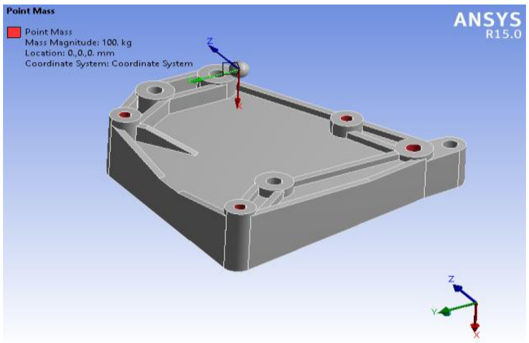


Fig-7: Load applied

Boundary conditions: Remote displacement with all translations and rotations fixed at the highlighted surfaces in the figure.

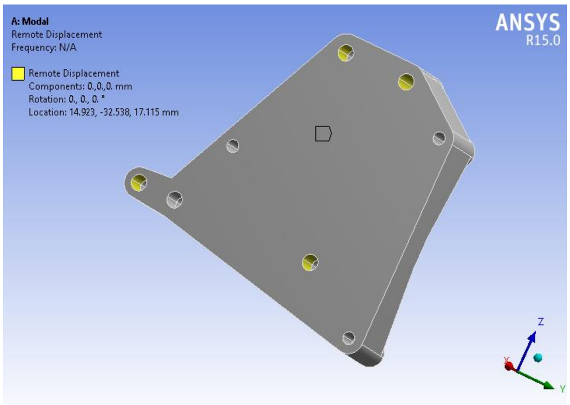


Fig-8: Boundary condition applied in the structure

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.

5. 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. It starts with the validation of the model. The frequencies for the 8 modes are calculated from the FEA results and are tabulated below.

Tabular Data

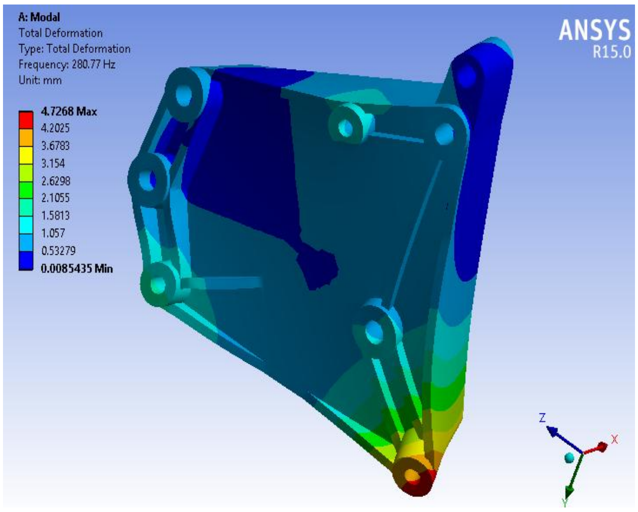
	Mode	Frequency [Hz]
1	1.	280.77
2	2.	356.51
3	3.	781.97
4	4.	4292.9
5	5.	4542.3
6	6.	6231.5
7	7.	8131.6
8	8.	8717.7

Table-2: Frequencies

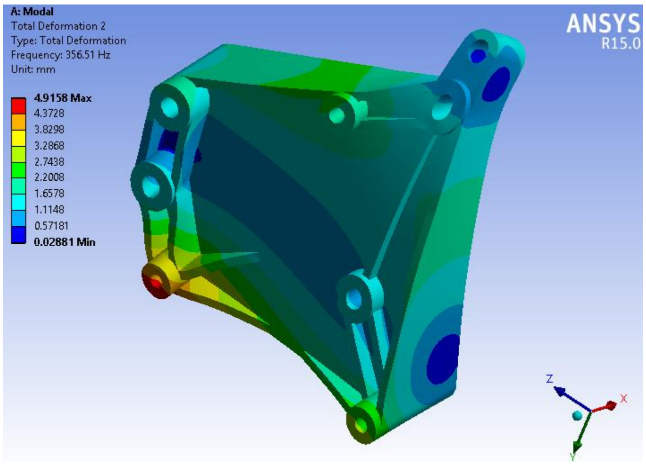
ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Total Deformation: the deformation a formed based on the different frequencies calculated from FEA.

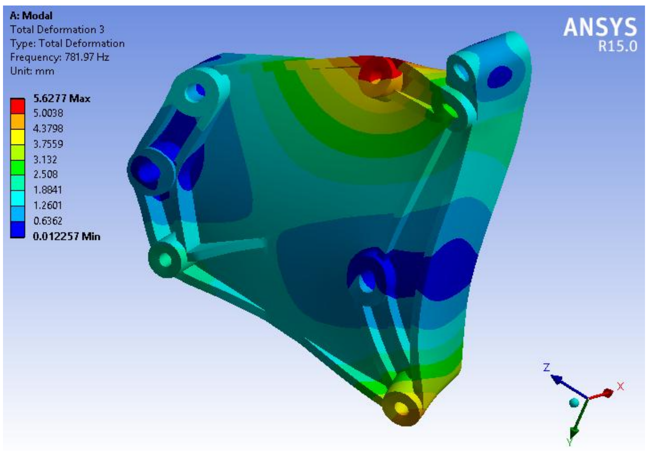
1st Mode:



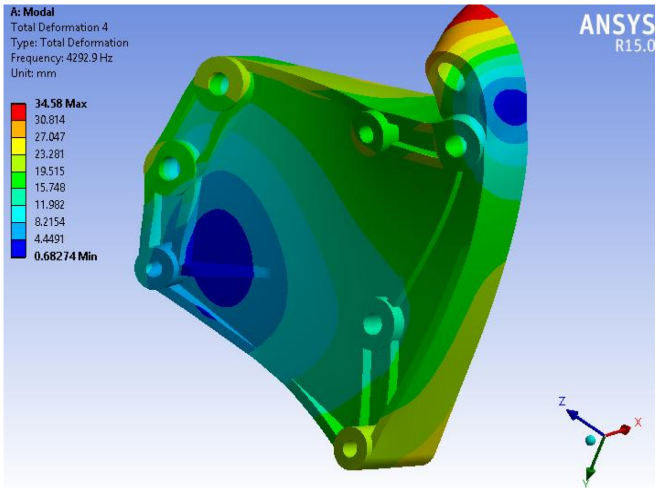
2nd Mode:



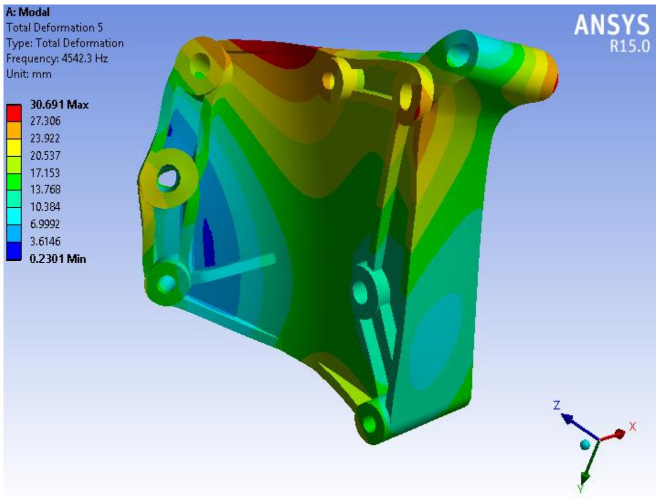
3rd Mode:



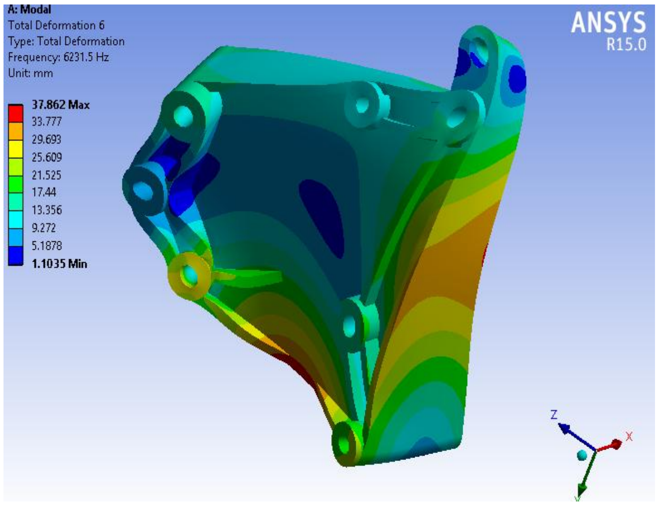
4th Mode:



5th Mode:

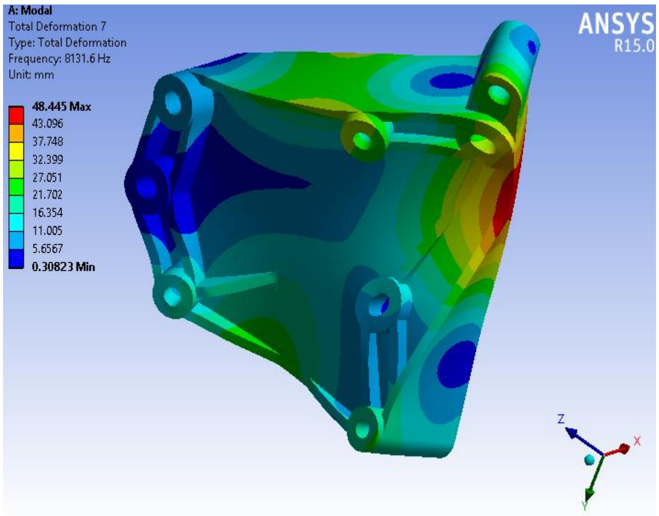


6th Mode:



ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

7th Mode:



8th Mode:

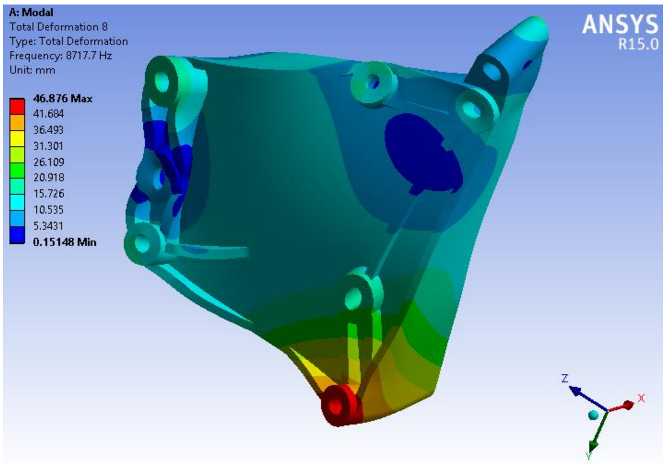


Fig-9: Total deformation _ True scale

6. CONCLUSION

The operating frequency range for the mounting bracket is 25Hz-75Hz. The first fundamental frequency calculated through dynamic analysis for the designed mounting bracket is 281 Hz which is above the operating frequency range and hence the design is safe.

7. REFERENCES

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

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

[3] 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.

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

[5] Jeong Woo Chang and Young Shin Lee (2008), "Topology optimization of compressor bracket", Journal of Mechanical Science and Technology, Vol. 22, 1668-16

[6] Asheber Techane (2007), "Dynamics and Vibration analysis of bus body structures", Proceedings of MSC/NASTRAN conference, Brazil.

[7] 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, 895-911.

[8] 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, 1667–1672.

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

[10] 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.

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