International Journal of Mechanical and Industrial Engineering

Volume 3 | Issue 1 Article 1

July 2013

# FINITE ELEMENT ANALYSIS AND NATURAL FREQUENCY OPTIMIZATION OF ENGINE BRACKET

UMESH S. GHORPADE
*Rajarambapu Institute of Technology, Sakharale, Islampur, India*, usghorpade@yahoo.com

D. S. CHAVAN
*Rajarambapu Institute of Technology, Sakharale, Islampur, India*, D.S.CHAVAN@gmail.com

VINAAY PATIL
*Vaftsy CAE, Pune, India*, vinaaypatil@vaftsycae.com

MAHENDRA GAIKWAD
*Vaftsy CAE, Pune, India*, MAHENDRA@gmail.com

Follow this and additional works at: https://www.interscience.in/ijmie

Part of the Manufacturing Commons, Operations Research, Systems Engineering and Industrial Engineering Commons, and the Risk Analysis Commons

Recommended Citation
GHORPADE, UMESH S.; CHAVAN, D. S.; PATIL, VINAAY; and GAIKWAD, MAHENDRA (2013) "FINITE ELEMENT ANALYSIS AND NATURAL FREQUENCY OPTIMIZATION OF ENGINE BRACKET," *International Journal of Mechanical and Industrial Engineering*: Vol. 3 : Iss. 1 , Article 1.
DOI: 10.47893/IJMIE.2013.1116
Available at: https://www.interscience.in/ijmie/vol3/iss1/1

This Article is brought to you for free and open access by the Interscience Journals at Interscience Research Network. It has been accepted for inclusion in International Journal of Mechanical and Industrial Engineering by an authorized editor of Interscience Research Network. For more information, please contact sritampatnaik@gmail.com.

# FINITE ELEMENT ANALYSIS AND NATURAL FREQUENCY OPTIMIZATION OF ENGINE BRACKET

**UMESH S. GHORPADE[1], D. S. CHAVAN[2], VINAAY PATIL[3] & MAHENDRA GAIKWAD[4]**

[1&2]Mechanical Engineering Dept., Rajarambapu Institute of Technology, Sakharale, Islampur, India
[3&4]FEA (Finite Element Analysis) Dept., Vaftsy CAE, Pune, India
E-mail : usghorpade@yahoo.com[1], vinaaypatil@vaftsycae.com[3]

**Abstract -** The automotive engine mounting systems are very important due to different aspects of vehicle performance. Early in improvement the building of the engine mounting system should be rapidly checked and precisely analyzed, without sample of a vehicle authorization. Engine bracket has been designed as a framework to support engine. Vibration and fatigue of engine bracket has been continuously a concern which may lead to structural failure if the resulting vibration and stresses are severe and excessive. It is a significant study which requires in-depth investigation to understand the structural characteristics and its dynamic behavior. This paper presents and focuses on some Finite Element (FE) analysis of a typical engine bracket of a car will be carried out and natural frequency will be determined.

***Keywords -*** *FEA;Modal FEA; Engine Mounting Bracket;*

## I. INTRODUCTION

Engine is one of the most important components of a road vehicle such as car. High performance sports car has their engine supported by bracket. It plays an important role in improving the comfort & work environment of a car. The improvement of engine bracket system has been the subject of intense interest for many years. It is necessary to design proper engine bracket for a car. As such, engine bracket has been designed as a framework to support engine. Vibrations and fatigue of engine bracket has been continuously a concern which may lead to structural failure if the resulting vibrations and stresses are severe and excessive. Prolonged exposure to whole-body vibration in the working environment may lead to fatigue and in some cases it damages the car.

Generally, the most important vibration relevant excitations in a car engine can be identified as follows:- combustion force; main bearing reaction forces including mass forces damper function and flywheel whirling, modified by the front-end damper; piston side forces including secondary motion; camshaft bearing reaction forces including mass forces, opening and closing impacts and bearing impacts; valve opening and closing impacts; valve train forces caused by chain/belt movement or gear drive; gear train forces inside the transmission; drive train reaction forces and moments. It is well-known from basic Non-linear vibration theory; improvement in the vibration control can be achieved by determining the natural frequency of the engine bracket system well below the frequency band in which excitation exhibits most of the vibratory energy. It is in this context, the development of engine bracket can make the engine capable of absorbing vibration. Automotive engine mounting system must satisfy the primary tasks such as engine movement, engine rigid-body dynamic behaviour, and vibration isolation. The design and development of mounting bracket through use of Ansys software to achieve the requirements for mounting system. Limits over the development of the mounting systems due to drivability and NVH concerns, provides savings in design resources. NVH is an important vehicles characteristic motivating to achieve overall customer satisfaction. Engine is mostly mounted to the front sub frame and once installed in a vehicle, engine mounting has a significant task in decisive the vehicle vibration characteristics. Optimizing the mounts system in early stages of engine design is possible by implementation of computer-aided engineering (CAE) tools. CAE results can be analyzed without any costly prototypes. The results can be used to define strategy for the vehicle mount system and optimize the locations and the rates of mounts. A good mounting system separates engine input vibration from the vehicle body and suppress the effect of road inputs to the vehicle driver.

## II. BRIEF OVERVIEW OF SOME RESEARCH

Zhang Junhong et al. [1] have investigated that vibratory and acoustic behaviour of the internal combustion engine is a highly complex one, consisting of many components that are subject to loads that vary greatly in magnitude and which operate at wide range of speed. CAE tools development will lead to a significant reduction in the duration of the development period for engine as well as ensure a dramatic increase in product quality. CAE capabilities in the simulation of the dynamic and acoustic behaviour of engine and focuses on the relative merits of modification and full-scale structural/acoustic optimization of engine, together with the creation of new low-noise designs. Modern

CAE tools allow the analysis, assessment and acoustic optimization of the engine.

Gabriel-Petru ANTON et al.[2] have investigated the NVH test-calculation correlation, the finite element (FE) model updating of an engine and the vibration level (low and medium frequency range) on the engine/body interface points . The main objective for this approach is to obtain the absolutes values of the vibration level (low and medium frequency range) on the interface points using an updated FE model. Experimental and theoretical analysis used for this work, have allowed us to understand the real vibratory behavior and to obtain a new FE model more closed by reality. The final updating, the test-calculation correlation results and also the operational simulation.

Senthilnathan Subbiah et al. [3] have developed durability tests on vehicles in the end-user environment to reduce failures and warranty costs in the end-user hands. The failure analysis of muffler mounting brackets of three-wheeler vehicles observed during the durability test. Cracks at the weld location between the engine cradle and brackets were observed in all the vehicles at an average distance of 10,000 km. Many possible causes of the failures are identified using fishbone diagram. Further investigations were carried out on the design using finite element method (FEM). A FEM model was developed for the engine cradle assembly in which engine and muffler were modeled as point mass. Results show high magnitude of stresses and strain energy at the weld location. Analysis of the design suggests that bracket was acting as a cantilever beam with one-plane welding mounted on the engine cradle. Modified design, though eliminated the above failure, shifted the failure mode to the bush-bracket region.

Youngwoo Choia et al. [4] have studied the measured acceleration signals are analyzed virtual damage and frequencies by extending the 160,000 km. For the accelerated vibration test, the acceleration test of six degrees of freedom (DOF) in the time domain using MAST and the acceleration test of single-DOF in the frequency domain using single axis equipment are performed. The measured PG acceleration signals as same as virtual damage of the 160,000 km, are used for the accelerated test of six DOF. The axis which shows the maximum virtual damage value from real road test is used for the single-DOF test. The tests are performed by enveloping and amplifying PSD lines of real roads after analyzing frequencies. No failure is found in the six-DOF acceleration tests. However in single-DOF tests, the connection part between the battery and bracket had been broken. It is the reason why the chosen test axis is with the maximum virtual damage values in analyzing the real roads. The results are satisfactory safety wise, however further research will improve optimization.

S.K. Loh et al. [5] has discussed about the focuses on some Finite Element (FE) analyses performed such as frequency analysis to determine the structural response due to harmonic excitation over a frequency range. The resonant frequency can be predicted based on the responses in frequency domain. Besides that, the static and dynamic vibration analyses give the maximum structural stress condition under static loading and dynamic condition. The predicted maximum stresses are compared with inherent material yield strength. The plastic deformation is not covered in the study as only elastic property is defined. A fatigue failure prediction of the current P-TAC motor bracket using FE simulation and fatigue failure criteria approach has also been studied. The dynamic stress curve giving mean stress and alternating stress has been applied in the established fatigue failure criteria such as Yield Criteria and Fracture Criteria to predict the possibility of fatigue occurrence. This approach is considered a conservative prediction approach to prevent structural fatigue which is best suited and safe for certain design applications. In the effort to strengthen the motor bracket, different modifications of the motor bracket geometry have been investigated and the comparison of results of each analysis is presented. The rib support, edge radius and thickness have been modified and added to examine the effect to the overall static and dynamic behavior. It is found that the rib-support addition and the increase of edge radius can effectively improve the structural performance based on the analyses involved.

S. Irving et al.[6] has investigated the fatigue performance of two different bracket connections for use in high-speed ocean craft. Constant amplitude, cyclic tests revealed that weld quality within the curved or nested insert has a profound effect upon the fatigue behaviour. Under severe conditions, the loss in fatigue performance due to poor weld quality may override the gain achievable by more optimal bracket designs. Under the condition of a good quality butt weld with deep penetration, the nested bracket has an improved fatigue performance when compared to the traditionally used soft toe bracket.

Mehmet Firat et al. [7] investigated that proposed approach is based on numerical simulation of stamping processes by using explicit – incremental and implicit – iterative finite element techniques. The influence of the numeric model parameters are investigated with factor analysis and described with response surfaces obtained by multi-linear regression. A forming process leading to spring back-critic channel geometry is selected for the application of the proposed methodology. The effects of modeling parameters are determined by evaluating influences of the punch velocity and the element size, in order to obtain a numerically calibrated simulation model. Then the sensitivity of the spring back deformations to the contact interface friction and the blank holder force is predicted, and a set of response surfaces is generated. Comparisons with the experimental data indicated the suitability of the proposed approach in spring back predictions. The proposed technique is employed in the stamping analysis of an engine suspension bracket made of high-strength steel.

**Fig. 1: Engine mounting bracket of a car**

## III. TYPES OF BRACKET

*A. Engine Mounting Bracket of Car*

The engine mounting bracket of the car as shown in the fig. 1 is the bracket which supports the engine from the back side. It is made of steel and made by the process of stamping. It is connected to the engine with large face and connected to the vehicle structure with the smaller end to take the load and absorb the vibrations. Its operational life is more because of the low vibration rate and even the knocking rate of the engine is less. But if the engine is old or there are some other problems related with the vehicle structure, then there are large chances of failure of the engine mounting bracket. The failure of the engine mounting bracket is mainly due to cracking which generates at the point where the stress level is high and propagates through the structure of the bracket. The stresses in the structure of the bracket are generated mainly due to engine vibrations and motion of the vehicle on uneven road surface.[10]

*B. Aeroplane engine's continental engine mounting bracket*

A mounting bracket is used as a base member having a flat upper surface and an elongated shoulder extending upward from the base surface. The shape of the bracket is v-shaped along its length. The base member has a tapped bolt hole adjacent at the base surface. The mounting bracket comprises of bracket member having an upper surface adapted to support a component, a flat lower bracket surface. The base is connected to the plane structure and the other part connected to the engine which takes most of the load. It is made in aluminium by casting process to make it light in weight and efficient. It is checked at regular interval to eliminate chances of major accidents. Also it has to take large amount of vibrations from the aero engine which works at the highest speed and generates the huge power. Due to such considerations, designing the bracket for these kind of engine is a difficult process. Fig. 2 shows Aeroplane engine's continental engine mounting bracket. [10]

**Fig. 2 : Aeroplane engine's continental engine mounting Bracket**

## IV. LOCATION OF BRACKET INSTALLATION

The engine mount assembly includes a support member arranged to be attached to a vehicle frame component. a pair of elastomeric engine isolators positioned relative to each other on the support member, and a pair of engine mounting brackets arranged to be positioned on the isolators and attached to a engine component, where in the engine isolators are arranged to maintain the engine mounting brackets in spaced relation to the support member thereby dampening engine vibration and controlling engine movement relative to the vehicle. A engine mounting system is often a primary path for noise. The vehicle's structure at the mounting location is crucial in regard to noise transmission, durability and crash worthiness. The upper bracket attaches to the transmission by use of two horizontal fasteners. A limited vertical space is available for both the rubber mount and bracket. With limited rubber volume it is not possible to allow the rubber to be as soft as it is desired for maximum isolation of vibration. Therefore the bracket must be designed to be as stiff as possible. Through the use of Computer Aided Engineering, an optimized bracket design for stiffness, strength and mass is obtained while still supporting a shortened development cycle. The primary functions of engine mounting systems are to support and position the engine in the vehicle, to isolate engine-generated vibrations, and to control external-to-vehicle induced vibrations and motion. engine mount brackets must support the engine (withstanding stress, temperature, and corrosion as the essential product, requirements), and are not to decrease the mount performance thus meet first mode resonance and stiffness requirements. The supporting bracket supports a support member which is provided on the engine. The retaining bracket is mounted on the vehicle body. In such an automobile engine-supporting structure, a cut-out portion is formed at either a bolt insertion-hole of the supporting bracket or a bolt insertion-hole of the retaining bracket for permitting the engine to be turned on an axis extending in a transverse direction of the vehicle in proportion with impact loads. The impact loads are imposed on the engine upon a collision of the vehicle. As a result, the engine is prevented from horizontal movement in a rearward direction of the vehicle upon a frontal collision of the automobile. [10]

**Fig. 3 : Engine Bracket**

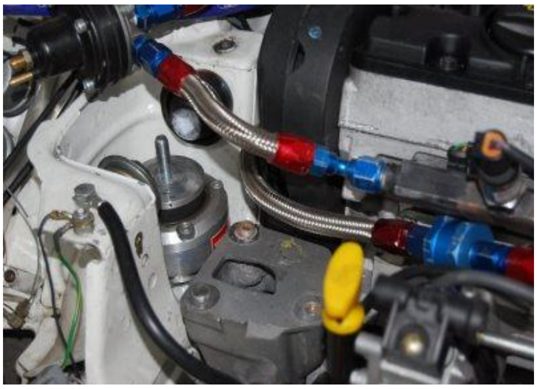

**Fig. 4 : Position of Engine Bracket.**

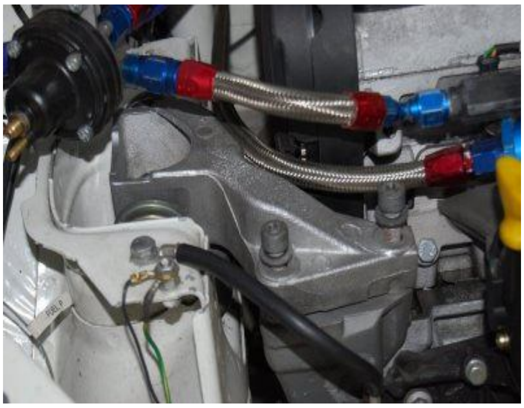

**Fig. 5 : Mounted engine bracket.**

The bracket and bush are assembled together; the bush is brought into engagement with the cut-out portions that are formed at the supporting arm portions of the bracket. The bush is then fixedly positioned between the supporting arm portions by means of the fixing bolt. [10]

## V. ANALYTICAL METHOD TO DETERMINE NON LINEAR VIBRATIONS

Consider the conservation system defined by the equation

$$\square + f(x) = 0 \qquad (1)$$

Now acceleration,

$$\square = Vdv/dx \qquad (2)$$

Substituting equation '2' in equation '1' we get

$$Vdv = - f(x)dx$$

Where $\square$ = V. If x = X when V=0, its integral is

$$|V| = |dx/dt| = \sqrt{{}_x\int^X f(\lambda)d\lambda} \qquad (3)$$

The second integral yields

$$t - t_0 = {}_0\int^x (d\square / {}_x\int^X f(\lambda)d\lambda) \qquad (4)$$

Where $t_0$ is the corresponding to x=0 the eqn. 4 expresses time as a function of displacement and its inverse is the displacement – time relationship.

If the motion is periodic with period T, the time corresponding to the motion from x=0 (t = $t_0$) to x=X(v=0) represent a quarter period. Hence we have

$$T = 4\ {}_0\int^x (d\square / {}_x\int^X f(\lambda)d\lambda) \qquad (5)$$

Thus the period becomes a function of the amplitude X.

## VI. THE FEM MODEL AND RESULTS

Finite element analysis (FEA) is one of the most popular engineering analysis methods for Non linear problems. FEA requires a finite element mesh as a geometric input. This mesh can be generated directly from a solid model for the detailed part model designed in a three-dimensional (3D) CAD system. Since the detailed solid model (see Fig. 6) is too complex to analyse efficiently, some simplification with an appropriate idealization process including changing material and reducing mesh size in the FE model is needed to reduce the excessive computation time. The engine mounting brackets are made of up Aluminium alloy or Mg alloy or Gray C.I. For thin bodies, a different type of meshing approach is required. For engine mounting bracket part, we extracted and meshed the mid-surface using Hex Dominant Quadrilateral and Triangular elements. [3]

Fig. 6 shows the FEM model of the existing design. The existing design has 4 holes. One hole is fixed and remaining three have force of 1000 N. This force is produced by Thrust. There is also self weight (g). The material used for FE Analysis is Non Linear. The FEM Model having 6 freedoms: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes.

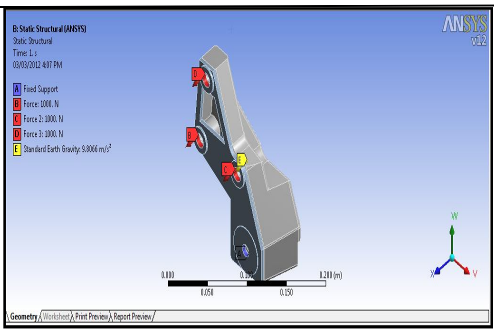


**Fig. 6 : FE Model of Mounted engine bracket.**

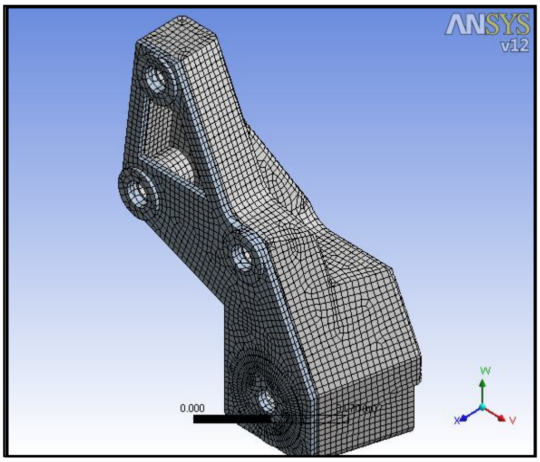


**Fig. 7: FE Mesh Model of Mounted engine bracket.**

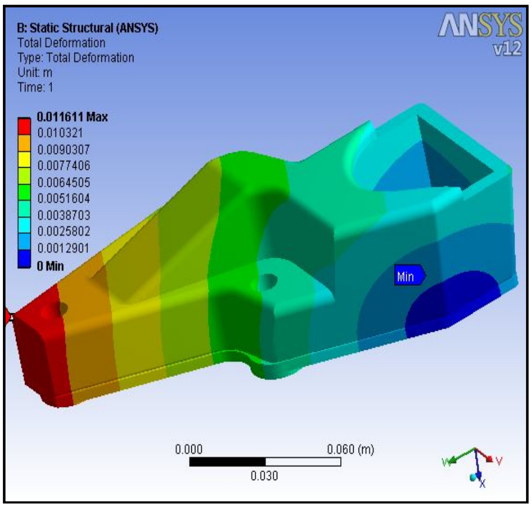


**Fig. 8 : Total Deformation of 3mm Mesh size Aluminium Alloy Mounted engine bracket.**

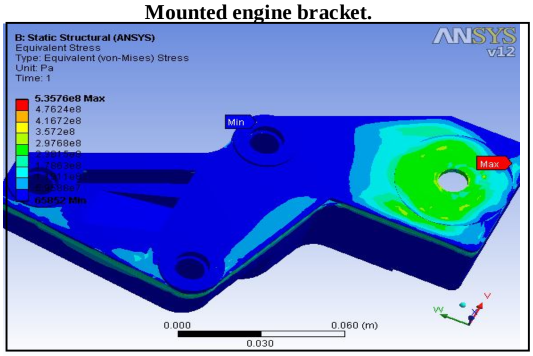


**Fig. 9 : Equivalent (von misses) stresses of 8mm Mesh size Aluminium Alloy Mounted engine bracket.**

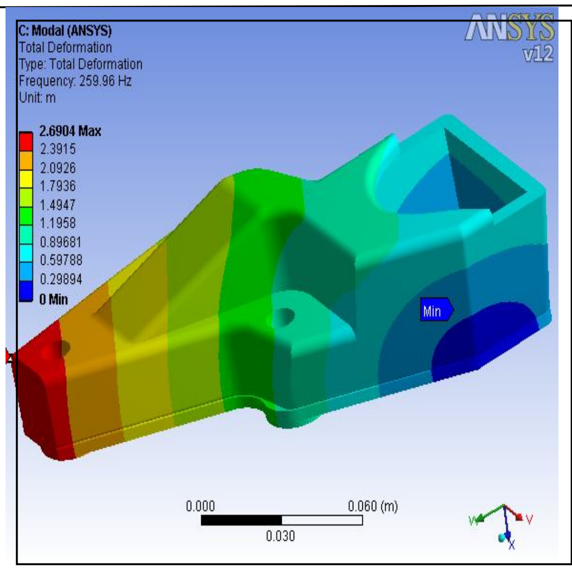


**Fig. 10 : Natural Frequency of 8mm Mesh size Aluminium Alloy Mounted engine bracket.**

**Result Table**

| **Sr. No.** | **Results** | | | |
|---|---|---|---|---|
| | ***Material*** | ***Max. Deformation (M)*** | ***Max. Stress (MPa)*** | ***Natural Frequency (Hz)*** |
| 1 | Al alloy | 0.011611 | 535.76 | 259.96 |
| 2 | Mg alloy | 0.027192 | 366.58 | 257.95 |
| 3 | Gray C.I. | 0.0032237 | 1495.2 | 198.75 |

## VII. CONCLUSION

Vibration plays a critical role in Engine components, especially in the supporting bracket.

Gray Cast Iron is essentially a brittle material and this is evident in the results that the low natural frequency will prove as a hindrance in vibration characteristic of the bracket.

In terms of analysis, Al alloy and Mg alloy are showing almost same value of natural frequency and indicate that any one of them would be a better choice than Gray Cast Iron.

However, in terms of FEA there is a caveat, which being that in Modal FEA, the effect of Damping is not considered. In Practical terms, Mg alloy exhibits better damping characteristics than Al alloy. Hence as far as the recommendation goes, Mg alloy will be preferred.

## VIII. ACKNOWLEDGMENT

We would like to acknowledge the role of Mr. Rahul Lawand, of Vaftsy CAE for his guidance in some aspects of ANSYS solutions.

## REFERENCES


[1] Zhang Junhong, Han Jun "CAE process to simulate and optimise engine noise and vibration" Mechanical Systems and Signal Processing 20 (2006) 1400–1409.

[2] Gabriel-Petru ANTON, Mihai PAVAL, Fabien SOREL, "APPLICATION ON AN UPDATED FINITE ELEMENT MODEL OF AN ENGINE IN THE AUTOMOTIVE INDUSTRY" SISOM 2011 and Session of the Commission of Acoustics, Bucharest 25-26 May.

[3] Senthilnathan Subbiah, O.P. Singh, "Effect of muffler mounting bracket designs on durability", Engineering Failure Analysis 18 (2011) 1094–1107.

[4] Youngwoo Choia, Dohyun Jungb, Kyoungchun Hamc, "A study on the accelerated vibration endurance tests for battery fixing bracket in electrically driven vehicles", Procedia Engineering 10 (2011) 851–856.

[5] S.K. Loh a, W.M. Chin a, Waleed F. Faris, "Fatigue analysis of Package Terminal Air Conditioner motor bracket under dynamic loading", Materials and Design 30 (2009) 3206–3216.

[6] S. Irving *, F.Ferguson-Smith, X.Z. Hu, Y. Liu, "Comparative fatigue assessment of soft toe and nested bracket welded aluminium structures", Engineering Failure Analysis 12 (2005) 679–690.

[7] Mehmet,OsmanH.Mete,UmitKocabicak,MuratOzsoy,"Stamping process design using FEA in conjunction with orthogonal regression", Finite Elements in Analysis and Design 46 (2010) 992–1000.

[8] Doo-Ho Lee,Jeong-Woo Chang,Chan-Mook Kim,"Optimal Shape Design of an Air-Conditioner Compressor Mounting Bracket in a Passenger Car," SAE International, 2003-01-1667.

[9] Hong Suk Chang, "A Study on the Analysis Method for Optimizing Mounting Brackets," SAE International, 2006-01-1480.

[10] Mahendra M. Gaikwad, "Bracket Document 112978," Coventry Engineers, UK.


◈◈◈