IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

Root Cause Analysis and Corrective Action Through Design Modification For Oil Filter Support Bracket

Dikshant Dinesh Nalawade[1], Sanjay Yadav[2], Aatmesh Jain [3], Jitendra Kumar [4]

[1]Student, [2]Professor, [3]Professor, [4]Professor

[1] Department of Automobile Engineering,

Abstract: The CAE tools are used in this work to assess and optimize the designs produced within the CREO software, which is used for the optimization of the oil filter support bracket used in our vehicles that are malfunctioning on the field. Finite Element Analysis was used to validate the bracket's endurance and other characteristics. The bracket is subjected to both static and dynamic analysis, as well as mode and stress analysis. The new design is optimized using form and topology optimization methods to reduce the breakage failure of the Fuel filter support bracket and increase the stiffness of the bracket. The optimized design generated by the CAE tool is then empirically verified. The new design results in tension levels that must be lower than the material's durability limit, add to greater mass while costing less, and is efficient in mounting assembly. The CAE tool findings had a very high match with the testing results**.**

***Index Terms* - oil filter, CAE tool , CREO software, optimization, fuel filter method.**

I. INTRODUCTION

The creation and development of an automobile is a complicated job for a business because it must meet all of the requirements for customer satisfaction as well as the various standards implemented by various authorities in various nations. It is a very market-oriented job with constant and immediate input from customers. One of the most essential components of a road vehicle, such as an automobile, is the oil filter. Fuel filters in high-performance cars are held by brackets. It is crucial in enhancing the comfort and working atmosphere of a vehicle. For many years, there has been a great deal of interest in improving the oil filter support mounting mechanism. A suitable oil filter support bracket for a car is required. The component producers have been under pressure to increase the performance of their materials and discover ways to supply them at lower costs due to the heavy emphasis on pricing. The body of the vehicle is impacted by a variety of sources of noise and vibration. The power provided across uneven surfaces, the engine, and the suspension produce a resonance effect over a wide frequency range, which causes the noise and vibration. Vibration from the engine and suspension transmitted to the body through the chassis mounting points has a substantial impact on a vehicle's ride and noise characteristics.

Press tool sheet metal processes are used to create the majority of automotive and aerospace components. These growing demands necessitate a design and production process based on predictions from the Finite Element Method for Sheet Metal Formability. The issue areas, causes, and remedies may be identified using computer-aided engineering techniques with the help of sheet metal's one-step formability analysis, and productivity can be increased by cutting down on expensive shop floor press tool trials and doing away with tool reworking. This article outlines the process by which the deformation of sheet metal components is affected by product design, metal characteristics, and production boundary conditions.

It is well known that body attachment stiffness plays a significant role in determining how quiet the vehicle is at idle and on the road. As a result, bracket was created as a structure to enable filter. Fatigue of the filter support bracket has long been a worry, potentially leading to structural failure if the ensuing stresses are extreme and excessive. Prolonged exposure to whole-body stressors in the workplace can cause fatigue and, in some instances, car harm. A modal investigation into the engine bracket is intended to ascertain whether the present configuration has a natural resonance frequency that is lower than the engine bracket's excitation the rate, in which circumstance the design will be deemed safe. After the design's safety is confirmed, material optimization will take place. Optimization: The design's appropriateness for various materials, including aluminum and magnesium alloy, will be evaluated. The project's goal is to choose an appropriate substance and strengthen the bracket. We've gotten a report of a dipstick bracket breaking in the field. Fuel filter pipe support bracket failure was recorded in the field truck equipped with an MDE5 engine in the market. Oil filter support supports are one of the components needed for oil filter stability. The position of an oil filter support plate on an oil filter in conjunction with the motor. The examination of displacements, stresses, strains, and forces on a structure or component as a result of the application of a load is known as a static structural analysis. It is presumed that the reaction and loads of the structure will change gradually over time. In this study, other forms of loading, including externally applied forces, pressures, and temperatures, can be used. Either a linear or nonlinear study of static structures is possible. The motor is immediately connected to the oil filter support bracket, which is affixed with the oil filter. Many shocks from the motor are transferred to the oil filter via a bracket. The vibrational properties of a building or specific component are identified through modal analysis in the form of natural frequencies and mode shapes. We may perform further in-depth dynamic analyses using this technique, such as transient dynamic analysis, harmonic analysis, or spectrum analysis. When designing a structure for dynamic loading circumstances, consideration must be given to the inherent frequencies and mode shapes. Only linear behaviour is valid in this study. In modal analysis, applied loads and damping are not taken into account. Pre-stressed modal analysis must be preceded by a static structural study. With the use of finite element methods, several technical and automotive components are analysed. In the post-processing stage, these analyses are highly useful for approving or making some design adjustments. The product life cycle mostly determines design adjustments, which aid design engineers or analysts in determining the final dimensions and component materials. A significant degree of vibration is experienced by an oil filter support bracket. Because of these movements, it is difficult to reduce the mass and redesign the oil filter support bracket while taking into account the material characteristics, boundary conditions, static and dynamic properties of the bracket.

II. OBJECTIVE

The goal is to perform a modal analysis on the oil filter support bracket to see if the present design has a natural frequency that is lower than the excitation frequency of the support bracket, in which case the design is deemed secure. Once the design's safety and tension level are determined, material optimization will take place. Optimization: The design will be evaluated for various materials, steel alloys, and material suitability and durability limits. FE evaluation It will be carried out to determine whether the collapse of the bracket was caused by motor movements. An FE study is conducted to validate the better bracket's endurance due to motor vibrations. To connect FEA, it is suggested to quantify accelerations at bracket mounting sites. Assuming that the bending procedure does not result in considerable leftover stress, the updated bracket meets the fatigue strength criterion.

1. To build and optimise an oil filter bracket in order to prevent failure.
2. Conduct FE analysis for engine vibration loads to determine whether bracket failure is due to elevated stress induced by engine vibrations.
3. The bracket construction is altered to decrease the stress level in the bracket as a result of motor vibration.
4. To reduce the likelihood of fuel filter support clamp breaking.
5. To assess the effectiveness of a changed oil filter attachment
6. Determine the oil filter assembly's lowest natural frequency of oscillation.

III. PROBLEM STATEMENT

In this project, I am using CAE tools to assess and optimize designs produced within the CRIO software, which is used for the optimization of the oil filter support bracket used in our trucks that are failing on the field in the South African market. Finite Element Analysis will be used to validate the bracket's endurance and other characteristics. A problem with the oil filter support bracket has been reported in the field.

In the current project, I'll be utilizing CAE tools to examine and improve the designs made in CRIO software, which is utilized to improve the oil filter support bracket used in our vehicles that failed in the field in the South African market. Finite Element Analysis will be used to confirm the bracket's durability and other characteristics. The bracket will be subjected to static and dynamic analysis, along with modal and stress evaluations.

The new design is optimized utilizing form and size optimization techniques to reduce the breaking failure of the Fuel filter support bracket and boost the rigidity of the bracket. The CAE tool's optimised design is then empirically confirmed. The new design leads to stress levels that must be within the material's limit of endurance, contributes to lower bulk and cheaper cost, and is useful in mounting assembly. The CAE tool's output and the experimental output exhibited excellent correlation.

The frame will be subjected to both static and dynamic analysis, as well as mode and stress analysis. The new design is optimized using form and size optimization methods with the goal of minimizing the breakage failure of the Fuel filter support bracket and increasing the stiffness of the bracket. The optimized design generated by the CAE tool is then empirically verified. The new design causes stress levels to be lower than the material's endurance limit, contributing to lower bulk and lower expense, which is successful in mounting construction. The CAE tool findings had a very high match with the testing results.

IV. METHODOLOGY

We conduct modal analysis on the oil filter support frame. Modal analysis is an essential instrument for comprehending component vibration properties. The motor retaining bracket is first hypothetically examined. The main purpose of the bracket is to hold the motor. To modify the existing engine holding bracket design and perform static structural analysis on various materials. Alloys include Gray C.I., Aluminum alloy, VSHR metal alloy, and ERW-1. Modal study of a modified engine mounting bracket shows that the component natural frequency is lower than the self-excitation frequency. The finest material was selected for the motor mounting bracket. The amplitude stress level at the failure region is around 130 MPa, according to the modal analysis findings. For the bracket substance VSHR170A, there is no fatigue data (Allowable stress limit) to establish whether it satisfies the fatigue strength standard. Due to a dearth of proof, it is difficult to attribute the bracket failure to the increased stress level at this location produced by motor vibration. It could, however, be one of the reasons of bracket failure. The oil filler tube, electrical harness, and fuel line all add significantly to the strain

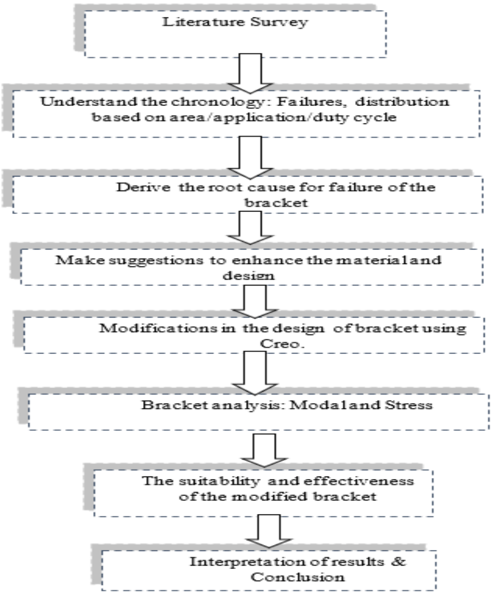


Fig 1. Flow Diagram

The majority of commercial aluminum alloys have much greater specific strengths than steel while having an aluminum density that is only around one third that of steel. Reduced vehicle weight would lead to greater oil savings in addition to considerable emissions reductions. For these reasons, aluminum is preferred over steel for usage in automotive applications.

However, there are a number of barriers to the use of aluminum in the automobile sector. Among them are,

1. Aluminum has substantially lower forming limitations than steel. There are several opportunities for aluminum to shred at bends. The forms that may be produced are constrained by this, thus die design, die testing, and applications are slowed down.
2. Thicker aluminum sheets have higher spring back issues, and maintaining dimensional standards is challenging.
3. Due to the heavier steel, the high cost, and the longer development time for stamping tools, traditional vehicle body production technology that uses stamped sheet steel.components cannot be properly developed to meet future vehicle needs.

With the aid of the FEA PACKAGE programmed, the engine mounting bracket is analyzed. First, in this procedure, the material choice is made using the FEA PACKAGE software's engineering data sheet. The software's MESH function is used to mesh the component after that. From the perspective of putting the results into practice, the meshing is crucial to the entire analytical process. In every FEA programmed, the size and form of the meshing determines the approach taken to solve a FE issue. By refining, which is nothing more than altering the mesh's size, one may achieve accurate results.

The big size end of the bracket that is fastened to the vehicle body is where the stress study of the engine mounting bracket is done. To do a static structural analysis, the load is then applied to the bracket's other end

V. RESULTS AND DISCUSSION

Fatigue data (Allowable stress limit) for the bracket material VSHR170A is not available to check whether it is meeting the fatigue strength requirement due to these we need to change the bracket material we need the material with high tensile strength and yield strength with exact allowable stress limit For oil filter bracket we use material from steel for cold forming group, which is durable, corrosion resistance, high sustainability, non-combustible, available in the form of sheet or strip.

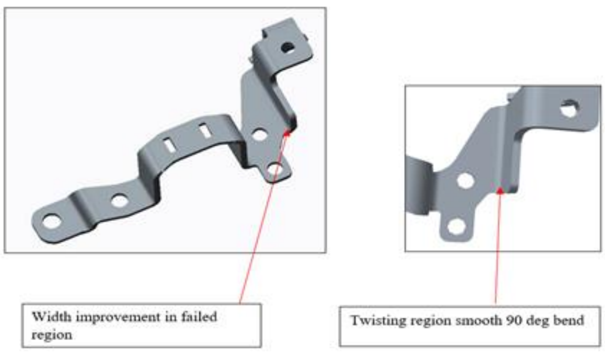
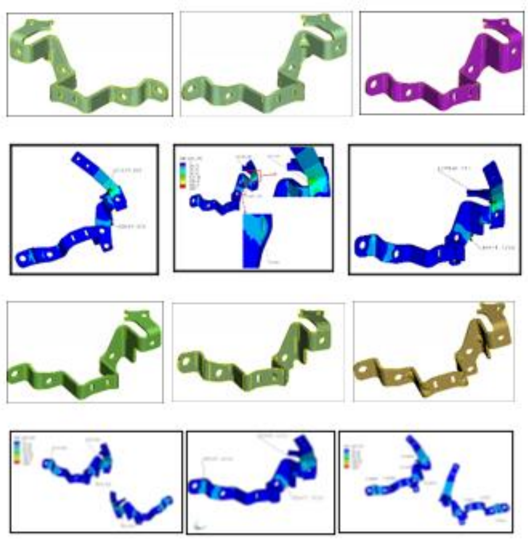


Fig.2 proposed CAD geometry with improved width and twisting region 90 deg bend

As per required conditions we select VSHR350 material for replacement of VSHR170A material, which is thermochemically rolled. The engine mounting bracket has been examined utilizing Package's finite element analysis tool. This effort makes a contribution to the creation of new engine mounting bracket material.

The static structural and modal analysis findings have demonstrated that VSHR metal is superior than aluminum. According to the findings, the VSHR metal bracket is secure for the specified use. The VSHR metal engine mounting bracket's reduced weight is its key benefit. VSHR metal is an environmentally favorable substance since it can be recycled. The VSHR metal bracket has a longer

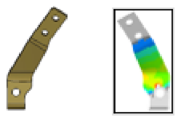
lifespan than an aluminum bracket and can be produced more quickly. As per proposed recommendations we must increase the width of failed region and thickness if there is any need. Twisting region changed to smooth 90 deg bend.
Addition of beads at bend locations due to spring back effect observed by supplier.



Iteration Number	Max stress value at failed region (MPa)	Max stress value at bracket (MPa)
1	88	129
2	87	128
3	84	131
4	62	102
5	78	94
6	63	86

Fig 3 Design iterations & stress analysis of oil filter support bracket

Continuous iteration helps to refine your product over a period. Instead of simply launching a product and releasing it into the world, continuous iteration aims to improve the product and keep up with the market.
VSHR metal brackets are more suitable for use as a bracket since they are less prone to corrosion. The biggest drawback of employing VSHR metal instead of aluminium is its greater price, however recent research has indicated that this cost differential is narrowing. Additionally, the cost of producing VSHR metal will undoubtedly go down as its application in industries rises.



Old filler bracket max stress (MPa)	New filler bracket max stress (MPa)
260	175

Fig 4 Design iterations & stress analysis of oil filler support bracket

Thus, it can be said that when choosing a material for an engine mounting bracket, VSHR metal might be favored over aluminum. It has also been demonstrated through research that VSHR metal engine mounting brackets perform better under various operating situations than aluminum brackets, which should be used instead.
It's a cost-effective approach which puts user experience at the heart of the design process
To reduce the stress level on failed region, according to all recommendations few design concepts are developed and analyzed'

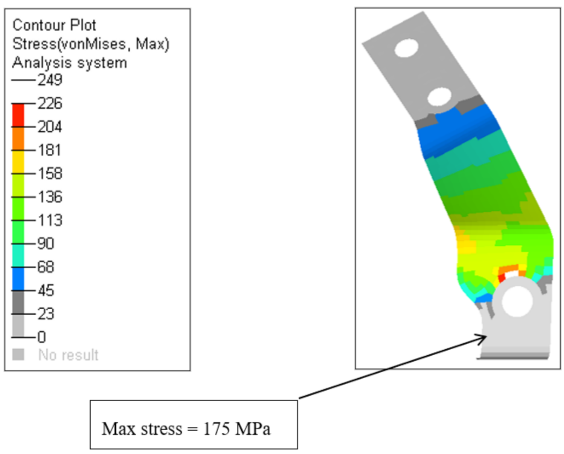


Fig.5 . Final Modification

For modified bracket

Endurance limit=0.5*420 = **210 MPa (Min)**

According to criteria stress level must be lesser than endurance limit, modified bracket max stress level is 175MPa while endurance limit is 210MPa, so the bracket pass in acceptable criteria.

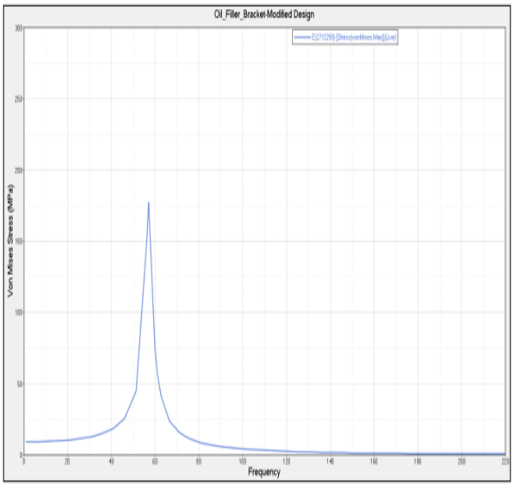


Fig.6 Frequency vs stress analysis of modified bracket

VI. Conclusion

First natural frequency of Current design and modified design are 51.1, 56.9Hz and in the range of Max torque frequency, there is chances occurrence of resonance.
From frequency response analysis, in current design stresses are exceeding the endurance limit and it is not meeting the durability requirement
From frequency response analysis, in new design stresses are below endurance limit and it is meeting the durability requirement
Bracket material is changed to VSHR350. Fatigue data for VSHR350 material is available. Allowable stress amplitude limit is 141 MPa.141 MPa is taken as allowable limit for VSHR350.
Stress level at failed location is reduced to about 63 MPa. Maximum stress level in the bracket is about 80 MPa. Bracket is made by blanking operation.
Assuming bending operation would not cause significant residual stress, this design can be considered as meeting the fatigue strength requirement.
New Design is meeting the durability requirement.

References

[1] Optimization of Compressor Mounting Bracket of a Passenger Car by Sachin Kalsi1 Daljeet Singh, J. S. Saini2
[2] Modal Analysis of Engine Mounting Bracket Using FEA by Yongkang Zhao
[3] Multi-axial Fatigue Analysis of Oil Suction Tube Bracket based on Multi Body
[4] Dynamics Solution of Inline 6-Cylinder Diesel Engine by Jozo SKOKO and Marko BASIC, Thomas RESCH, Brett SHEETS, Stefan BRUNNER
[5] Root cause analysis in quality problem solving of research information systems: a case study by Mohammad Javad Ershadi, Roozbeh Aiasi and Shirin Kazemi
[6] One Step Formability Analysis of Fuel Filter Bracket using Siemens NX by Chandranath.M, Dr. D. Ramegowda and Dr G Mallesh
[7] Simulation and experimental validation of powertrain mounting bracket design obtained from multi-objective topology optimization by Qinghai Zhao, Xiokia chen.
[8] Topology Optimization of Front Leaf Spring Mounting Bracket by Vijay Kalantre, K. H. Munde, Ashish Pawar.
[9] A failure analysis method for designing highly reliable product-service systems by Koji Kimita, Tomohiko Sakao and Yoshiki Shimomura.
[10] An Improved Method of Failure Mode Analysis for Design Changes by Rafael Laurenti, Henrique Rozenfeld
[11] Finite Element Analysis of Fuel Tank Mounting Bracket by Abhishek P, Hardeep S M, Sarvocch G, Anil L.
[12] NPTLE (Product design and manufacturing) Course Calculation instruction: NVH and dynamic fatigue analysis, Magnus Gustafsson Material standard STD-311-0003