ISSN: 2235-767X
Volume 07 Issue 05
May 2019

ACKNOWLEDGEMENT

I am privileged to have got an opportunity to submit this project stage-I report " Engine mounting system topology optimization using FEA and Experimental stress analysis Technique."

I take this opportunity from bottom of my heart and express a deep sense of gratitude towards my seminar Guide **Prof. Yogesh Ingulakar** for guiding me and standing by my side whenever I faced any kind of difficult.

I would also like to thank the head of department **Prof. Yogesh Ingulakar**, who gave me the opportunity to work on such a challenging seminar.

I am extremely thankful to **Dr. Nihar Walimbe**, Principal of Dhole Patil College of Engineering, Pune, for providing me infrastructure facility to work in, without which this work would not have been possible, I thank them from my bottom of my heart.

Jagtap Kundlik Gajanan

ABSTRACT

Engine mountings are widely used in automotive to hold Engine with chassis. Automobile industry regularly improving from many years with the efforts in modification of the mechanical parts of vehicle in order to improve their performance. The most effective way of increasing automobile mileage while decreasing emissions is to reduce vehicle weight. Existing mount has scope of mass optimization in current design. Finite element analysis of mounting will be done using Hypermesh and Ansys. Ansys software will be used for topology optimization. Fixture will be designed for mounting bracket on UTM. Experimental stress of mount will be measured using strain gauge and applying corresponding loading through UTM. Validation for strain

vector from FEA & Experimental results will be done. Conclusions and future scope will be suggested.

Engine mounting system topology optimization using FEA and Experimental stress analysis Technique.

INTRODUCTION

1.1 Background:

In this automotive area the need for light weight structural materials is increasing as there is a more focus on fuel consumption reduction and improvement in decreasing the emission. The magnitude of production volumes has traditionally placed severe requirements on the robustness of process used in the manufacturing. The manufacturers have strong importance on the cost has the demand for the component to improve the material performance and to deliver these materials at low cost is the requirement.

In automobile sector the extremely competitive automotive business needs manufactures to pay a lot of attention to traveling comfort. Resonant vibration is from unbalanced masses exist within the engine body, this is causing the designers to direct their attention to the event of top quality engine mounting brackets so as to confirm that there is improvement in riding comfort. The demand for higher play acting engine mount brackets should not be offset by arise within the production prices and/or development cycle time.

In diesel engine, the engine mounting bracket is the major problem as there is unthrottled condition and higher compression ratio and even there are more speed irregularities at low speed and low load when compared to gasoline engines. So due to this there are more vibration excitation. By this vibration engine mount bracket may fail, so by optimizing the shape and thickness of engine mount bracket we can improve the performance at initial design stages. By some studies it is observed that brackets saved 38% of mass. Structural optimization is an important tool for an optimum design; comparison in terms

ISSN: 2235-767X
Volume 07 Issue 05
May 2019

of weight and component performance structural optimization techniques is effective tool to produce higher quality products at lower cost.

1.2 Problem Statement

In this automotive era the need for light weight structural materials is increasing as there is a more focus on fuel consumption reduction and improvement in decreasing the emission. Currently, bracket contains excess material, leads to increase in weight of the vehicle. Directly affects the mileage and cost. In this thesis, modeling present bracket in CAD software and analyzing it for induced stresses and deformation in CAE software. Then using topology optimization material will be removed. Again, analysis will be done on optimized model for stresses and deformation. It is also tested experimentally and results were correlated it with analysis results.

1.3 Objectives

1. Modeling current bracket.
2. Analyzing for stresses and deformation.
3. Topological optimization for the model.
4. Analyzing for stresses and deformation.
5. Experimental testing and correlating results.

2. LITERATURE REVIEW

Umesh S Ghorpade, D.S Charan, Vinay Patil and Mahendra Giakwad [1], in this paper they have designed engine mount bracket of a car and focused on to determine natural frequencies of car engine mount bracket. They have considered the three materials for engine mount bracket that is aluminum alloy, magnesium alloy, gray cast iron when modal analysis is carried out, it is found natural frequencies of gray cast iron is low which will prove more hindrance in vibration of engine mount bracket so they have eliminated gray cast iron, in terms of analysis aluminum alloy and magnesium alloy are showing almost near value of natural frequency in practical terms as magnesium alloy is having better strength that is low stress value, so preferably magnesium alloy is selected as better material by study.

Mr. Pramod Walunje, Prof. V.K. Kurkute [2], in this work they have mainly focused on the use of light weight material for bracket and also to reduce the weight of the bracket. Here the weight of the material is reduced and pre processing and post processing is carried out and even with this an experimental setup is also used to find the stress level of the materials they have observed that aluminum alloy have good natural frequency and stresses are also within the yield strength, so by considering the aluminum and reducing its thickness further by 2mm than original component, they found that now von misses stresses are also with in yield stress so they have achieved reduction in the mass of bracket up to 0.43kg when compared to previous one.

Dr.Yadavalli Basavaraj, Manjunatha.T.H [3], the analysis results which are obtained from two arm, three arm, four arm, filler arm and four arm symmetry engine mounts in which the design of four arm symmetry engine mount curve obtained from LS-Dyna approach follows exactly the experimental test curve and also this design has the highest natural frequency amongst all design iterations. The above work indicated that the rubber used in the engine mount had increased the frequency from 1.2Hz (basic design) to 1.8Hz (four arm symmetry). As the design is changing in rubber, the mode of frequency increases and it has found that 1.8Hz is the frequency for the four arm symmetry mount design. P.D. Jadhav, Ramakrishna [4], this work is a contribution to the development of new material for engine mounting bracket. The results obtained for the static structural and modal analysis have shown that the magnesium is better than aluminum. From the results it can be seen that the magnesium bracket is safe for the required application. The main advantage of the magnesium engine mounting bracket is its light weight. It will help in decreasing the weight of the power train assembly, which can increase fuel efficiency. The magnesium bracket can be manufactured with less amount of time and it posses longer life compared to an aluminum bracket. The magnesium bracket is less susceptible to corrosion; therefore they are better for the application of bracket. The main problem of using magnesium instead of aluminum is its higher cost; but recent studies have shown that the difference between costs of aluminum and magnesium is decreasing. Also if the use of magnesium in industries increases, its manufacturing cost will be definitely reduced. Thus it can be concluded that magnesium can be preferred over aluminum as a material for an engine mounting bracket. Research in the direction of implementing magnesium engine mounting bracket instead of aluminum bracket has also proved that magnesium brackets are better in various operating conditions.

Sandeep Maski, Yadavalli Basavaraj [5], as vibration and strength plays an important role in the design of engine mount bracket, so in this paper special attention has been given for selection of suitable material for engine mount bracket so that it can withstand high strength and vibrations. It can be seen from the obtained results and discussion that for modal analysis first fundamental natural frequency of mild steel is high that is 65 Hz compared with cast iron and wrought iron and in case of static analysis displacement of mild steel is 1.6 mm which is less when compared to other two materials and the maximum von-misses stress coming on mild steel is lesser than yield point, hence mild steel is considered as better material to design engine mount bracket.

B. Sreedhar, U. Naga Sasidhar [6], on topology optimized model finite element analysis has carried out for normal mode, FRF and strength analysis and complete parametric study has carried out by using Hyper Study. In this paper, topology optimization approach is presented to create an innovative design of an engine mount bracket. Final comparison in terms of weight and component performance illustrates that structural optimization techniques are effective to produce higher quality products at a lower cost.

3. ENGINE MOUNTING BRACKET

Engine mounts themselves are small parts that are meant to stabilize, as well as properly align, a vehicle's engine. So, even though these mounts are small, they play a large role in the overall functionality of the heart of your vehicle. Moreover, when these supposedly small and minor aspects of the vehicle go bad. An automotive engine-body-chassis system is typically subjected to unbalanced engine forces, uneven firing forces especially at the idling speeds, shaking forces and torques due to reciprocating parts, dynamic excitations from gearboxes and accessories, and road excitation. These tendencies give rise to undesired vibrations which lead to an uncomfortable ride and also cause additional stresses in the automobile frame and body. Vibrations are annoying and their origin can be difficult to detect. An engine mounting system includes a front mount, a rear mount, an engine mount, and a transmission mount. Installation heights and spring constants of each mount are predetermined such that the majority of the weight of the power train is supported by the front and rear mounts. Mounting axes of the engine mount, front mount, and rear mount are vertical while the mounting axis of the transmission mount is lateral. The transmission mount may include bridges which vary its spring constant during vehicle roll. There are many good reasons to resiliently mount an engine and/or transmission one increasingly important reason is to reduce structure bone noise and vibration generated by the engine and transmitted to the vehicles operator compartment. Resilient mounting will also provide longer life for frame and engine block mounting brackets, suspended components and transmission by attenuating transient shock inputs and operating torque loads. The paper also summarizes engine disturbances looks at several ideal practical mounting approaches and points out important limiting considerations. Finally, selection criteria and required data for proper engine mounting are outlined. A structure for mounting an engine for a vehicle comprising: a front engine mount for mounting the engine on a front side of the engine in a longitudinal direction of the vehicle, the front engine mount including an engine mount bracket which is fixed on one end thereof to a suspension member and has an insulator held on the other end thereof, the front engine mount mounting the engine with the insulator interposed between the engine and the engine mount bracket, the insulator supporting an engine bracket fixed to the front side of the engine, wherein a space is provided between the front side of the engine and the engine mount bracket; and an auxiliary equipment disposed in the space between the front side of the engine and the engine mount bracket, wherein the front engine mount has a strength against a load applied thereto in the longitudinal direction of the vehicle less than that of the auxiliary equipment. In this paper the front engine mount bracket as shown in figure 1 is taken under consideration.

3.1 Requirement Of Engine Mounting Bracket

The mounts can cause a variety of problems for your vehicle.

1. Engine Vibration

The first symptom to note is an excessive amount of engine vibration. Because motor mounts are meant to keep an engine secure, bad mounts will lead to an insecure engine that will bounce about. At times, there may be a sound emitting from the engine hinting of vibration, however, the more common symptom is a felt vibration on the passenger's side. If you do not often have visitors sitting on the passenger's side that can tell you something is wrong, place your hand on the passenger's seat from time to time to check for excessive vibration.

2. Misalignment

Securing a motor not only fastens an engine, but aligns it, meaning that the engine's height is ensured to be equal on all sides. If the motor mounts are in fact bad, the engine will sag and droop to one side. Again, there may be noises emitting from the engine that do not sound quite right. Inspect the engine to see if it is tilted. If so, your mounts aren't doing their job. However, if the engine is tilting excessively, an engine block may result. In other words, your car may stop in its tracks. Check the engine for drooping and, if so, replace any and all motor mounts as necessary.

3. Engine Damage

The third symptom is an extreme case. If motor mounts break off completely from an engine, and are not just loose or cracked, an engine can shift from one side to another, bouncing about. This presents a safety hazard if you are driving your vehicle in high speeds; the engine may shift and bounce about so violently that various parts will fly off.

4. Broken Belts and Hoses

Besides indirectly breaking engine parts, bad mounts can lead to damaged belts and hoses. Again, these various engine fan belts and radiator hoses will be damaged only during high speeds.

5. Excessive Noise

Earlier symptoms include parts that are simply not working, from engine parts to belts and hoses. However, the most common symptom is just excess noise. Note any knocking or clanking, as this general noise means something is wrong. It may mean one of the previous symptoms, or something completely new, but regardless of what the problem is, it means that you know something is wrong and to take your vehicle in for repairs immediately.

4. TOPOLOGY OPTIMIZATION

Topology optimization is a mathematical approach that optimizes material layout within a given design space, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance targets.

4.1 Basic Theory

There are three kinds of structure optimization,

- Size Optimization
- Shape Optimization
- Topology Optimization

Three optimization methods correspond to the three stages of the product design process, namely the detailed design, basic design and conceptual design. Size optimization keeps the structural shape and topology structure invariant, to optimize the various parameters of structure, such as thickness, section size of beam, materials' properties; shape optimization maintains the topology structure, to change the boundary of structure and shape, seek the most suitable structure boundary situation and shape; topology optimization is to find the optimal path of materials' distribution in a continuous domain which meet the displacement and stress conditions in structure, make a certain performance optimal. Thus, compared to size and shape optimization, topology optimization with more freedom degree and greater design space, its greatest feature is under uncertain structural shape, according to the known boundary condition and a given load to determine the reasonable structure, both for the conceptual design of new products and improvement design for existing products, it is the most promising aspect of structural optimization. For continuous structure topology optimization, there are some mature methods like: uniform method, evolutionary

ISSN: 2235-767X
Volume 07 Issue 05
May 2019

structural optimization method, variable density method etc. Uniform method introduced cell structure of micro structure (unit cell) in the elements of the structure, each unit cell has three forms, namely non-material voids (size = 1), isotropic-material entity medium (size = 0) and orthotropic-material opening-hole medium (0 < size < 1). Wherein the distribution of each form will be able to describe the form of topology and the shape of structure; evolutionary structural optimization method believe that stress in any parts of the structure should under the same level in an ideal structure. That means the local material with a low stress state is not fully utilized, so you can delete the material artificially. So gradually remove material which in a low stress state, and then delete the update rate, so optimized structure becomes more uniform. Variable density method is used to conduct optimization in this paper. The basic idea is to introduce a hypothetical material which density is variable and range from 0 to 1.

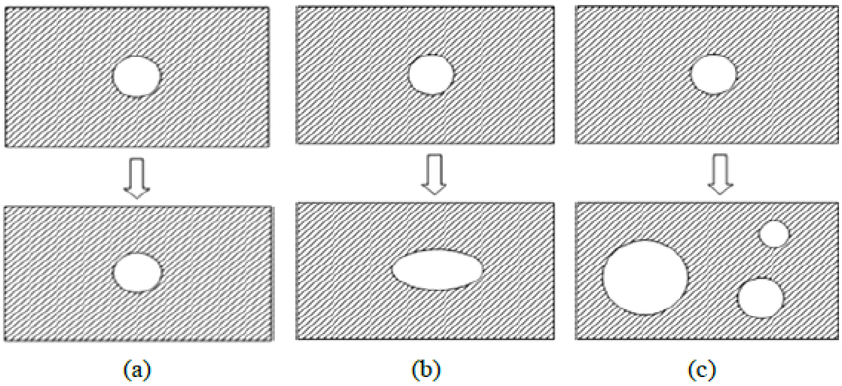


Fig.1 Three kinds of structure optimization. (a) Size optimization, (b) shape optimization, (c) topology optimization.

After changing the continuous structure to finite element model, then making per units' density as the design variables, to convert topology optimization problem into the optimal distribution of material, while in order to suppress the intermediate density of the material (material between void and entities), the introduction of a interpolation penalty factor used to describe the relationship between Young's modulus and density of the material which shown as the following formula:

$$E = x^p E_0 \qquad \ldots..(1)$$

ISSN: 2235-767X
Volume 07 Issue 05
May 2019

where: p—interpolation penalty factor ($p > 1$); E_0—Young's modulus of densified material unit; E—Young's modulus after interpolation. So that the original model with intermediate density will be eliminated or replaced by densified material and will achieve optimal results which close to the entity. Therefore, during topology optimization, variable is relative density of units, then structural topology optimization problem is converted into the optimal distribution of the material.

4.2 Process Of Topology Optimization

Based on Hyper Works platform topology optimization holder, first, according to the engine mounting position, we establish the three-dimensional geometric model of engine bracket, and then pretreated in HyperMesh, define design area, objective function and constraints under the optimization panel, finally operate topology optimization which design process is as Figure 2.

4.3 Analysis

To establish the geometry model by CATIA, then input the geometry to the ANSY to carry out pre-treatment operations like geometry cleanup, meshing, loads, constraints, etc.

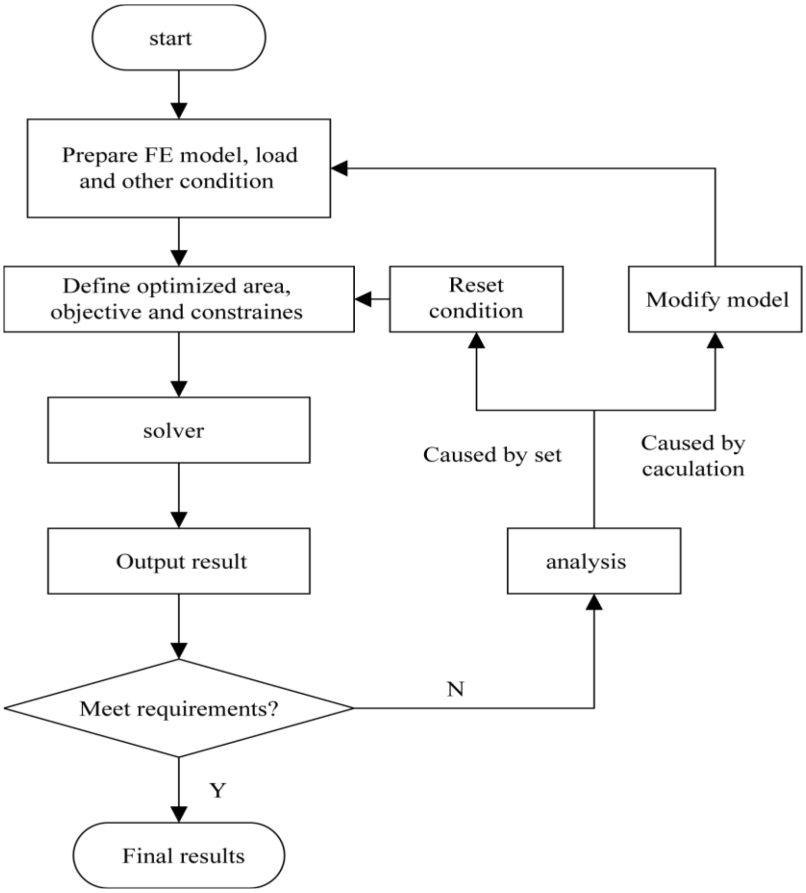


Fig 2. Process of Topology Optimization

Initially we need to collect the information regarding different loads acting on the bracket and the packaging data for fixing design space. The base bracket results from testing and finite element analysis (FEA) point of view for evaluating final optimized design.

4.4 Steps in Topology Optimization

The topology optimization consists of the following sequence of steps.

- Define the design space
- Define optimization parameters
- Material removal process and detail design

➢ Defining the Optimization Parameters

The aim of topology optimization in this project is to minimize the volume without affecting the bracket stiffness and strength compared to base bracket, so the design objective is taken as to minimize the volume. Following parameters are defined as constrains:

1. Allowable stress limit value is defined as stress constraints from durability point of view

2. Single draw direction is defined as manufacturing constraint.

- **Material Removal Process and Detail Design**

The optimization process took some iteration to remove the unnecessary material from the design space. The output of the topology optimization, an intermediate model which may be called a topology based model, is constructed by removing unnecessary materials from the rough conceptual model.

- Additive Manufacturing

Without manufacturing constraints, optimized components often have complex geometries. Recent developments in Additive Manufacturing (3D printing) enable the fabrication of these optimized designs without compromise. Topology optimization forms the natural design technology for Additive Manufacturing, as it fully exploits its potential.

- Shape Optimization

Where topology optimization excels at automated concept generation, shape optimization allows for efficient final fine tuning of designs. However, every fabrication technique has its shape/property accuracy limitations, particularly at the micro scale. Our activities on shape optimization concentrate on dealing with fabrication inaccuracy, and optimizing for robustness.

5. FEA OF ENGINE MOUNTING BRACKET

Engine mounting bracket of TATA INDICA is taken for analysis.

The Finite Element Method is a numerical approximation method, in which the complex structure is divided into number of small parts that is pieces and these small parts are called as finite elements. These small elements are connected to each other by means of small points called as nodes. As the finite element method uses matrix algebra to solve the simultaneous equations, so it is also known as structural analysis and it's becoming primary analysis tool for designers and analysts.

The three basic FEA process are

a) Pre processing phase

b) Processing or solution phase

c) Post processing phase

A static structural analysis is the analysis displacements, stresses, strains and forces on structure or a component due to load application. The structures response and loads are assumed to vary slowly with respect to time. There are various types of loading that can be applied in this analysis which are externally applied forces and pressures, and temperatures.

FEA Pre Processing For the Mounting Bracket:

The pre-processing of the engine mounting bracket is down for the purpose of the dividing the problem into nodes and elements, developing equation for an element, applying boundary conditions, initial conditions and for applying loads. The information required for the pre-processing stage of the bracket is as follows,

5.1 Geometry of The Engine Mounting Bracket

Dimensions of bracket are found physically with help of vernier caliper. Engine mount bracket is modeled with help of CATIA (V5) software (as shown in fig.5)

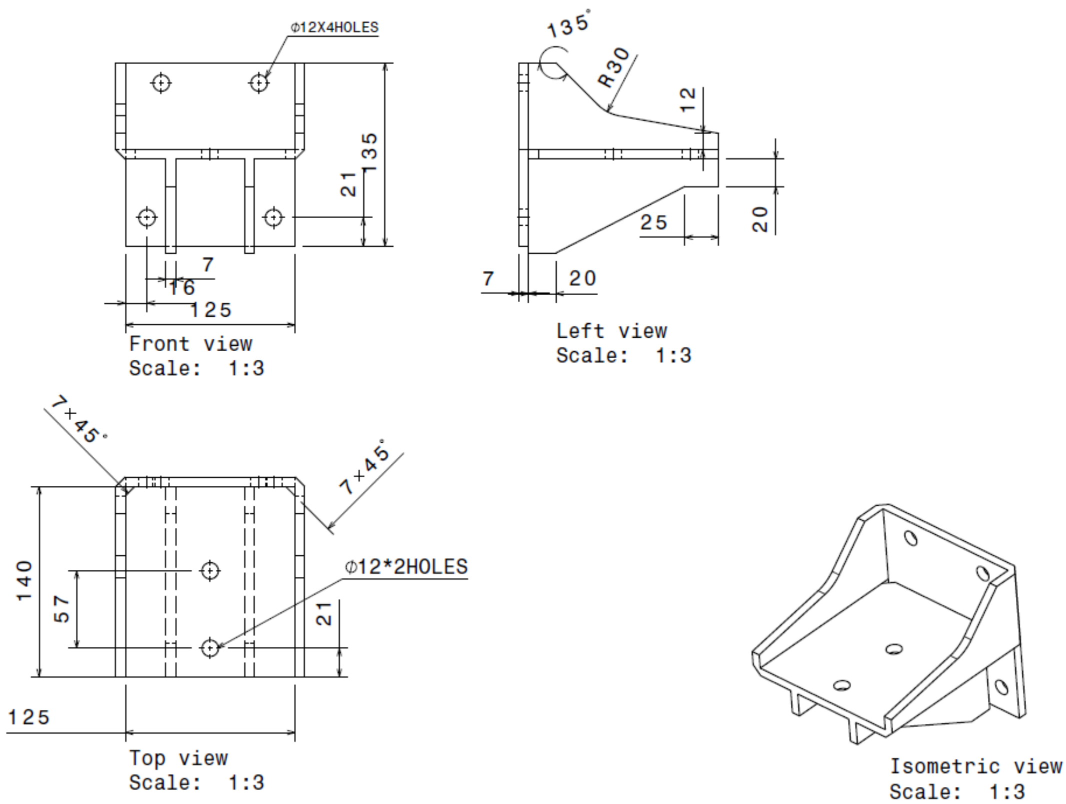


Fig.2 Drafting of Engine Bracket

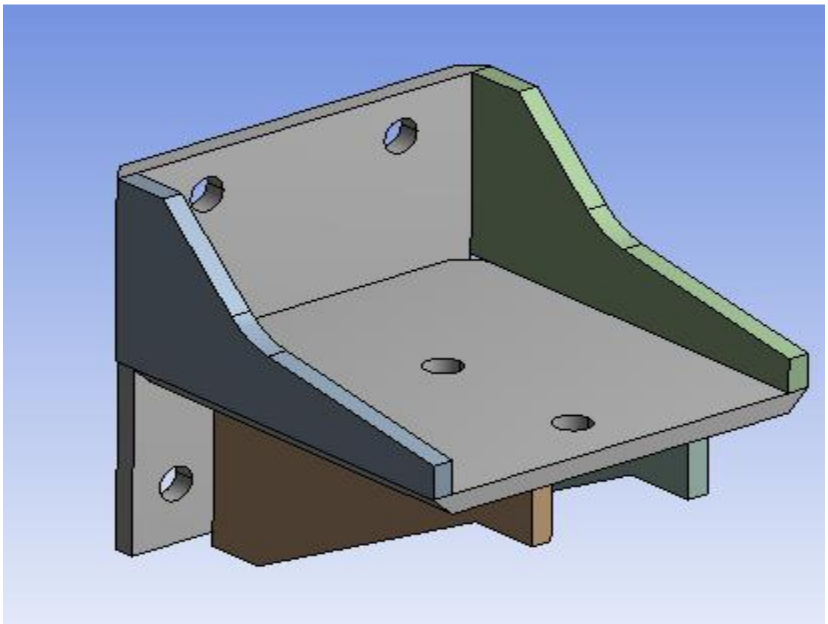
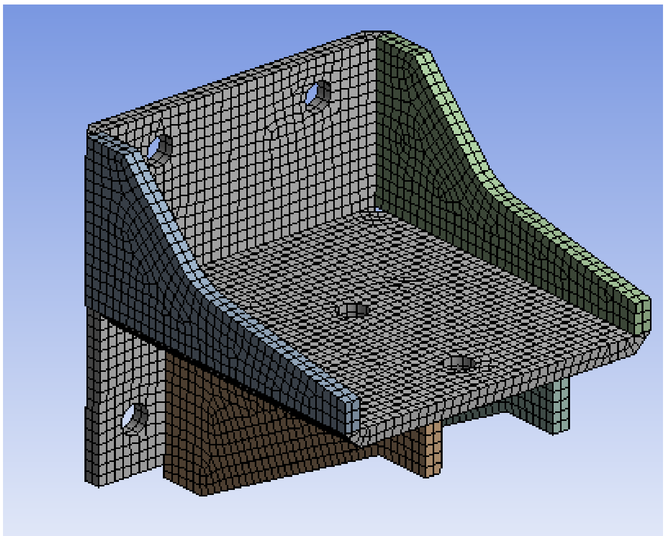


Fig.3 CATIA model

5.2 Mesh Generation

ANSYS Meshing is a general-purpose, intelligent, automated high-performance product. It produces the most appropriate mesh for accurate, efficient multiphysics solutions. A mesh well suited for a specific analysis can be generated with a single mouse click for all parts in a model. Full controls over the options used to generate the mesh are available for the expert user who wants to fine-tune it. The power of parallel processing is automatically used to reduce the time you have to wait for mesh generation.

Creating the most appropriate mesh is the foundation of engineering simulations. ANSYS Meshing is aware of the type of solutions that will be used in the project and has the appropriate criteria to create the best suited mesh. ANSYS Meshing is automatically integrated with each solver within the ANSYS Workbench environment. For a quick analysis or for the new and infrequent user, a usable mesh can be created with one click of the mouse. ANSYS Meshing chooses the most appropriate options based on the analysis type and the geometry of the model. Especially convenient is the ability of ANSYS Meshing to automatically take advantage of the available cores in the computer to use parallel processing and thus significantly reduce the time to create a mesh. Parallel meshing is available without any additional cost or license requirements.



Statistics	
Nodes	49490
Elements	11516

Fig. 4 Meshing of Engine Bracket

5.3 Boundary Conditions

• **5.3.1 Loading**

Loads: Specific values of load are implemented for a typical mounting bracket. The load is taken as 1000N. Load is applied at the two holes of the engine mounting bracket, which are connected to the engine structure with the help of rigid elements such as nut and bolts(as shown in fig.7)

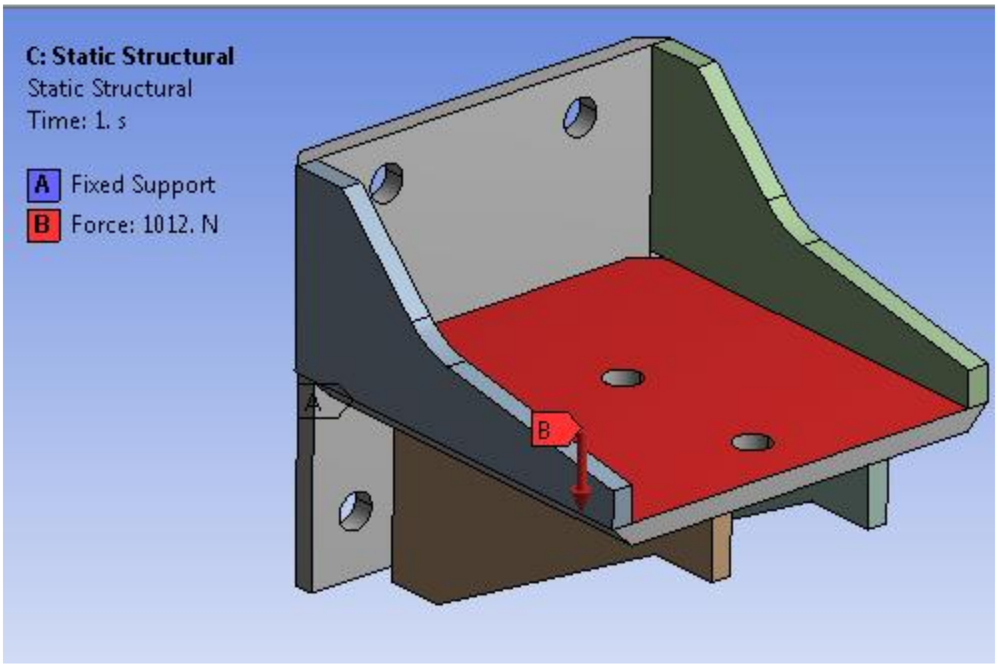


Fig.5 Boundary Condition to Engine Bracket

Total Deformation

The total deformation & directional deformation are general terms in finite element methods irrespective of software being used.

Directional deformation can be put as the displacement of the system in a particular axis or user defined direction.

Total deformation is the vector sum all directional displacements of the systems.

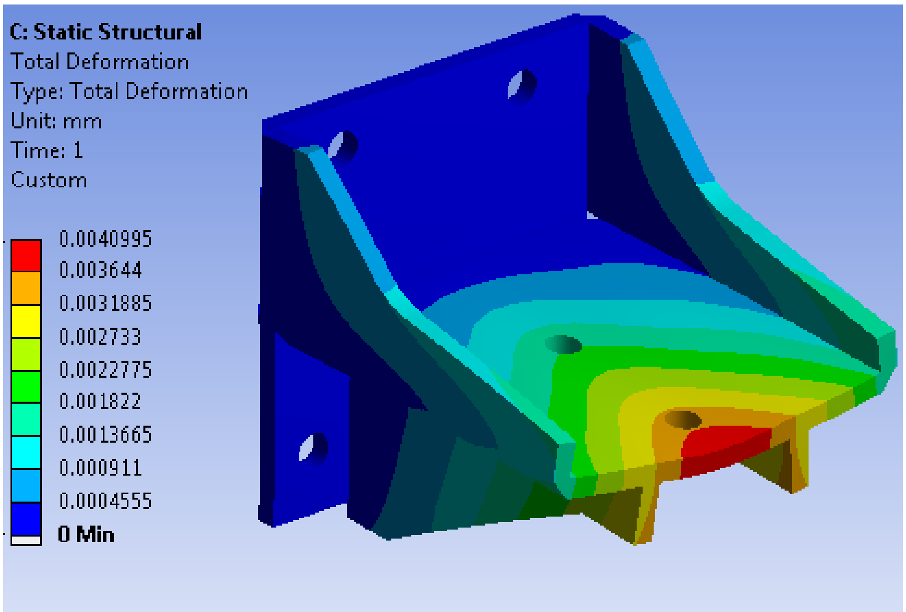


Fig.6 Total Deformation of Engine Mounting Bracket

Equivalent Stress

Equivalent stress is related to the principal stresses by the equation:

$$\sigma_e = \left[\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2} \right]^{1/2}$$

Equivalent stress (also called *von Mises stress*) is often used in design work because it allows any arbitrary three-dimensional stress state to be represented as a single positive stress value. Equivalent stress is part of the maximum equivalent stress failure theory used to predict yielding in a ductile material.

The von Mises or equivalent strain ε_e is computed as:

European Journal of Business & Social Sciences
Available at https://ejbss.org/
ISSN: 2235-767X
Volume 07 Issue 05
May 2019

$$\varepsilon_e = \frac{1}{1+\nu'}\left(\frac{1}{2}\left[(\varepsilon_1-\varepsilon_2)^2+(\varepsilon_2-\varepsilon_3)^2+(\varepsilon_3-\varepsilon_1)^2\right]\right)^{\frac{1}{2}}$$

Where:

ν' = effective Poisson's ratio

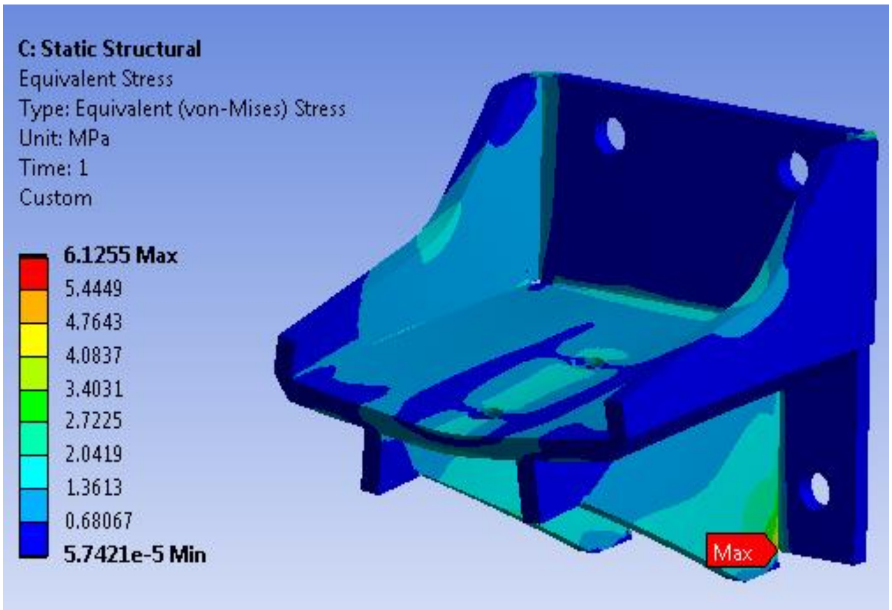


Fig.7 Equivalent stress on Engine Bracket

TOPOLOGY OPTIMIZATION:

European Journal of Business & Social Sciences
Available at https://ejbss.org/
ISSN: 2235-767X
Volume 07 Issue 05
May 2019

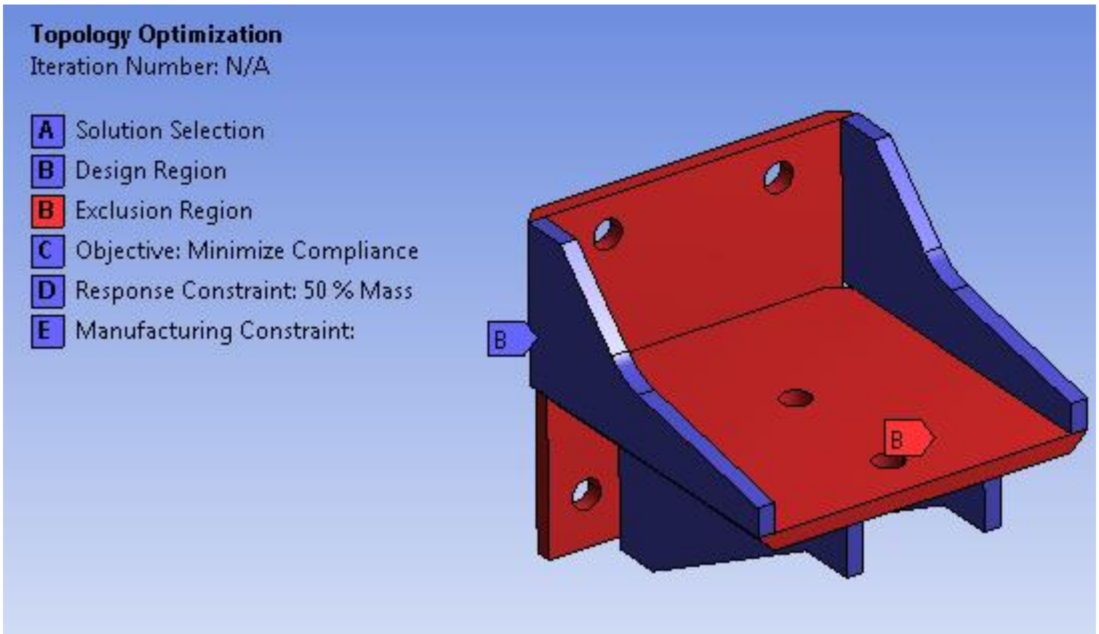


Fig.8 Topology Optimization

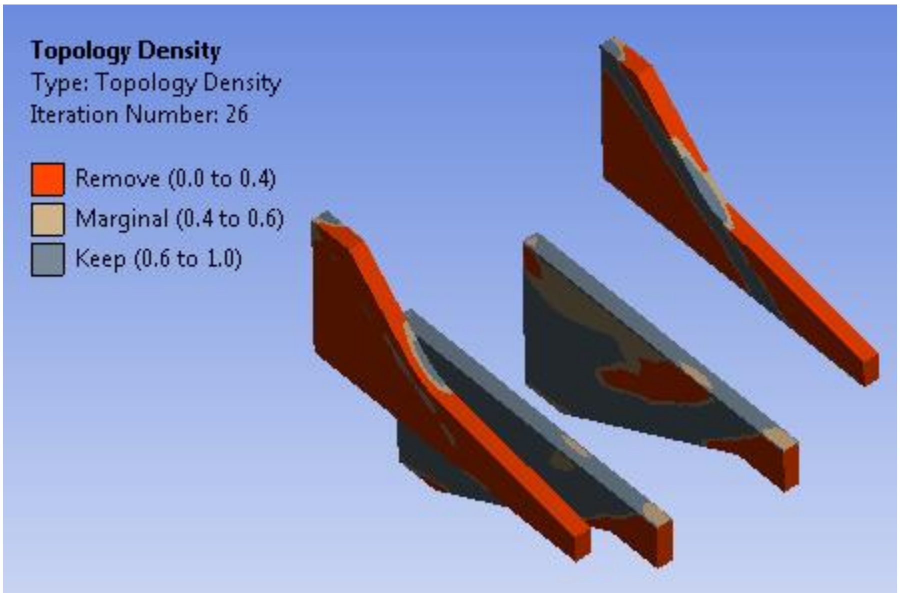


Fig.9 AREAS WHICH CAN BE USED FOR OPTIMIZATION

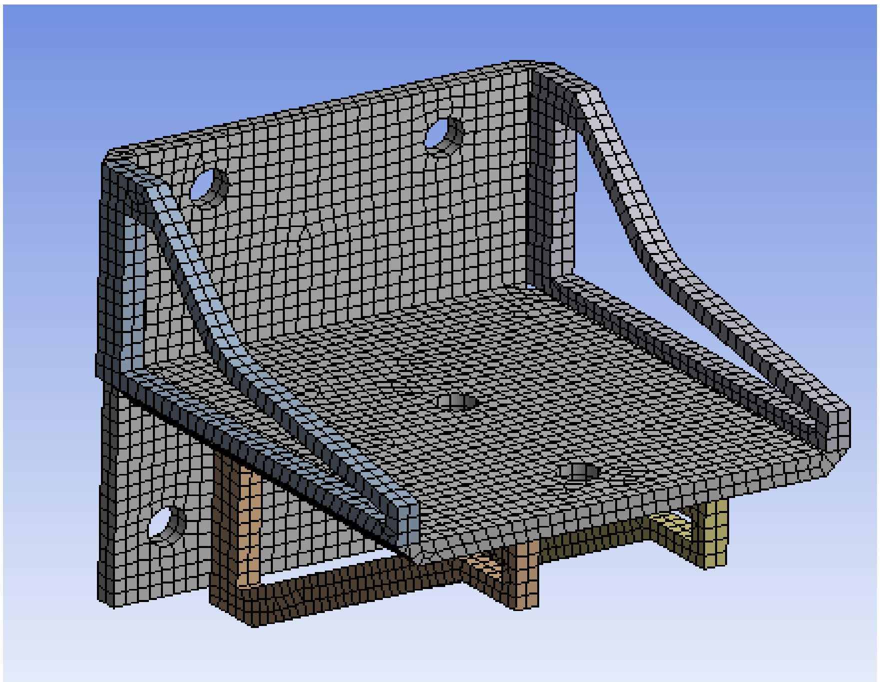
RESULTS OBTAINED AFTER TOPOLOGY OPTIMIZATION:

MESH

ANSYS Meshing is a general-purpose, intelligent, automated high-performance product. It produces the most appropriate mesh for accurate, efficient multiphysics solutions. A mesh well suited for a specific analysis can be generated with a single mouse click for all parts in a model. Full controls over the options used to generate the mesh are available for the expert user who wants to fine-tune it. The power of parallel processing is automatically used to reduce the time you have to wait for mesh generation.

Creating the most appropriate mesh is the foundation of engineering simulations. ANSYS Meshing is aware of the type of solutions that will be used in the project and has the appropriate criteria to create the best suited mesh. ANSYS Meshing is automatically integrated with each solver within the ANSYS Workbench environment. For a quick analysis or for the new and infrequent user, a usable mesh can be created with one click of the mouse. ANSYS Meshing chooses the most appropriate options based on the analysis type and the geometry of the model. Especially convenient is the ability of ANSYS Meshing to automatically take advantage of the available cores in the computer to use parallel processing and thus significantly reduce the time to create a mesh. Parallel meshing is available without any additional cost or license requirements.

ISSN: 2235-767X
Volume 07 Issue 05
May 2019



Statistics	
Nodes	42049
Elements	9095

Fig.10 Meshing of Engine Bracket After topology optimization

Boundary Condition

A boundary condition for the model is the setting of a known value for a displacement or an associated load. For a particular node you can set either the load or the displacement but not both.

The main types of loading available in FEA include force, pressure and temperature. These can be applied to points, surfaces, edges, nodes and elements or remotely offset from a feature. The way that the model is constrained can significantly affect the results and requires special consideration. Over or under constrained models can give stress that is so inaccurate that it is worthless to the engineer. In an ideal world we could have massive assemblies of components all connected to each other with contact

ISSN: 2235-767X
Volume 07 Issue 05
May 2019

elements but this is beyond the budget and resource of most people. We can however, use the computing hardware we have available to its full potential and this means understanding how to apply realistic boundary conditions.

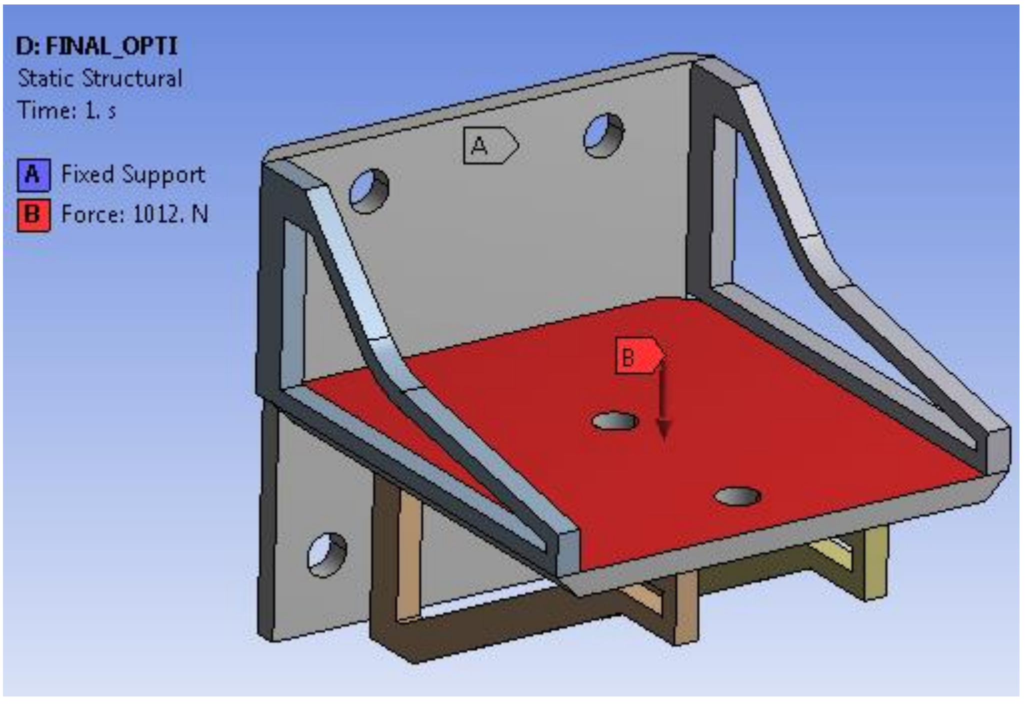


Fig.11 Boundary condition of Engine Mounting Bracket

Total Deformation

The total deformation & directional deformation are general terms in finite element methods irrespective of software being used.

Directional deformation can be put as the displacement of the system in a particular axis or user defined direction.

Total deformation is the vector sum all directional displacements of the systems.

European Journal of Business & Social Sciences
Available at https://ejbss.org/
ISSN: 2235-767X
Volume 07 Issue 05
May 2019

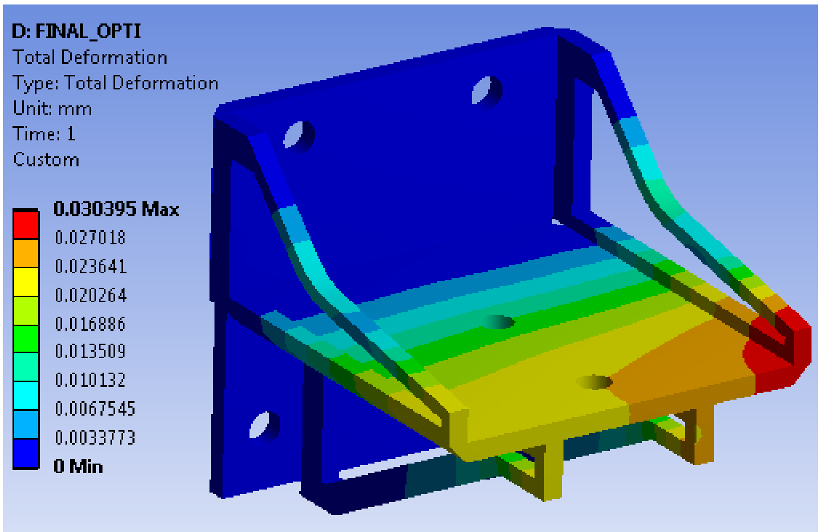


Fig.12 Total Deformation of Engine Mounting Bracket

Equivalent Stress

Equivalent stress is related to the principal stresses by the equation:

$$\sigma_e=\left[\frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2}{2}\right]^{1/2}$$

Equivalent stress (also called *von Mises stress*) is often used in design work because it allows any arbitrary three-dimensional stress state to be represented as a single positive stress value. Equivalent stress is part of the maximum equivalent stress failure theory used to predict yielding in a ductile material.

The von Mises or equivalent strain ε_e is computed as:

$$\varepsilon_e = \frac{1}{1+\nu'}\left(\frac{1}{2}\left[(\varepsilon_1-\varepsilon_2)^2+(\varepsilon_2-\varepsilon_3)^2+(\varepsilon_3-\varepsilon_1)^2\right]\right)^{\frac{1}{2}}$$

Where:

ν' = effective Poisson's ratio

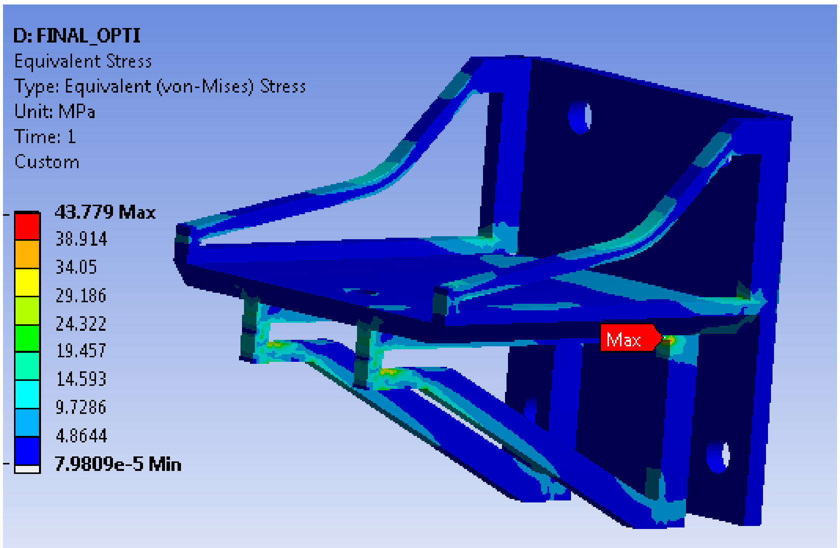


Fig.13 Equivalent Stress of Engine mounting Bracket

RESULT:

SR.NO.	CHARACTERISTICS	BRACKET BEFORE OPTIMIZATION	BRACKET AFTER OPTIMIZATION

ISSN: 2235-767X
Volume 07 Issue 05
May 2019

1.	Total Deformation(mm)	0.00409	0.03059
2.	Equivalent Stress (MPa)	6.1255	43.779

From the Analysis Results shown above it is clear that the Deformation of Engine Bracket after the optimization is less than that of the Original engine bracket.

REFERENCES:

1. Umesh.S.Ghorpade, d.s.chavan, vinay patil & mahendra gaikwad, "*Finite Element Analysis and Natural Frequency Optimization of Engine Bracket*" (IJMIE) ISSN No**.** 2231 –6477, Vol-2, Iss-3, 2012.
2. Mr.Pramod Walunje, Prof.V.K.Kurkute, "*Engineering Optimization of Engine Mounting Bracket Using FEA*" Indian Journal of Research ISSN-2250-1991 volume:2, Issue:12, Dec 2013.
3. Dr.Yadavalli Basavaraj, Manjunatha.T.H, *Design Optimization of Automotive Engine Mount System*, International Journal of Engineering Science Invention Issn (Online): 2319 – 6734, Issn (Print): 2319 – 6726, Volume 2 Issue 3, March. 2013
4. P.D. Jadhav , Ramakrishna , *Finite Element Analysis of Engine Mount Bracket*, Internation Journal of Advancement In Engineering Technology, Managament And Applied Science, Volume 1, Issue 4, Issue no. 2349-3224,Setember 14
5. Sandeep Maski, Yadavalli Basavaraj, *Finite Element Analysis of Engine Mounting Bracket by Considering Pretension Effect and Service Load*, IJRET: International Journal of Research in Engineering and Technology, e-ISSN: 2319-1163, p-ISSN: 2321-7308, Volume: 04 Issue: 08, August-2015.
6. B. Sreedhar,U. Naga Sasidhar, *Optimization of Mounting Bracket*, htc2012