OPTIMIZATION OF SILENCER MOUNTING BRACKET OF AN INDIAN COMMERCIAL LIGHT TRUCK

A dissertation report submitted in partial fulfilment of the requirement for the award of

MASTER OF ENGINEERING

IN

CAD/CAM ENGINEERING

Submitted By

SANCHIT SHARMA

Roll No. 801481020

Under the Guidance of

Mr. DALJEET SINGH

Assistant Professor

Mechanical Engineering Department

TU, Patiala.

MECHANICAL ENGINEERING DEPARTMENT

THAPAR UNIVERSITY

PATIALA-147004, INDIA

July 2016

DECLARATION

I hereby declare that the work in this dissertation report entitled **"OPTIMIZATION OF SILENCER MOUNTING BRACKET FOR AN INDIAN COMMERCIAL LIGHT TRUCK"** is an authentic record of my study carried out as a requirement for the award of degree of **Master of Engineering (CAD/CAM Engineering)** at **Thapar University, Patiala** under the guidance of **Mr. DALJEET SINGH,** Assistant Professor, Mechanical Engineering Department, Thapar University, Patiala, during **July 2015 to July 2016.** The matter embodied in this report has not been submitted in part or full to any other university or institute for the award of any other degree.

Sanchit
15/7/2016

(Sanchit Sharma)
Reg. No. 801481020

This is to certify that above declaration made by the student concerned is correct to the best of our knowledge and belief.

Daljeet Singh 15/7/2016

Mr. DALJEET SINGH
Assistant Professor,
Mech. Engg. Deptt,
Thapar University,

Patiala-147004.

Countersigned by:

Dr. S.K. MOHAPATRA
Sr. Professor and Head,
Mechanical Engineering Department,
Thapar University,
Patiala-147004.

Dr. S. S. BHATIA
Dean of Academic Affairs,
Thapar University,
Patiala-147004.

ACKNOWLEDGEMENT

I would like to express a deep sense of gratitude and thank profusely to my guide ***Mr. Daljeet Singh*** *for his sincere & invaluable guidance, suggestions and attitude, which inspired me to submit thesis report in the present form. Their dynamism and diligent enthusiasm have been highly instrumental in keeping my spirits high. His flawless and forthright suggestions blended with an innate intelligent application have crowned my task with success.*

I am also thankful to ***Dr. S.K. Mohapatra,*** *Sr.Professor & Head, Department of Mechanical Engineering for his encouragement and inspiration for execution of the thesis work.*

I am deeply indebted to my parents for their inspiration and ever encouraging moral support, which enabled me to pursue my studies.

I am also very thankful to the entire faculty and staff members of Mechanical Engineering Department for their intellectual support and cooperation.

Sanchit Sharma

Reg. No.: 801481020

ABSTRACT

With the increase in competition in the industries and new policies of government to reduce carbon emission from automobiles there is a need for timely and cost effective solution for the problems based on the design of any component. The different CAE tools can be useful for solving complex problems in simple ways.

In the present work, the CAE tools are used for the optimization of the silencer mounting bracket used in automobile. Both static and dynamic analysis was done on the bracket. With the objective to minimize the mass and increase the stiffness of the bracket, the new optimized design is made using different structural optimization techniques.

INDEX

LIST OF FIGURES

LIST OF TABLES

List of Symbol

Ω	=	Open bounded set (--)
Γ_N	=	Neumann boundary condition (--)
Γ_D	=	Dirichlet boundary condition (--)
A	=	Isotropic Elasticity Tensor, (psi)
$e(u)$	=	Strain tensor, (in/in)
u	=	Displacement field, (in)
f	=	Vector valued volume forces, (lb_f)
g	=	Surface loads, (lb_f)
ξ	=	Symmetric Matrix (--)
λ	=	Lame's first parameter, (psi)
I	=	Identify matrix, (--)
t	=	Time, (sec)
D	=	Working domain, (--)
V	=	Fixed Volume, (in^3)
Γ_0	=	Optimization Boundary
ℓ	=	Lagrange multiplier, (--)
n	=	Unit vector normal to surface, (--)
H	=	Mean curvature of boundary, (--)
Ψ	=	Level set function, (--)

CHAPTER 1 INTRODUCTION

1.1 COMPUTER AIDED ENGINEERING

Developed and developing nations are introducing new rules and policies which are aimed to increase the mileage of automobiles which will help in reducing the dependence of nation on foreign oil and crude. As there is shortage of petroleum in the world, we need to protect Earth from pollution and global warming so we have to reduce the consumption of fuel and this can be achieved in case of automobiles by reducing their weight. Moreover a product design and development is a tough task in this new competitive market of automobile industry. There is need of continual development and improvement in quality and to add more features with cost effectiveness is a challenge to organization. To reduce the cost, the design engineer always tries to minimize the usage of raw material, labour, machine and other costs that are essentially required to manufacture an automobile.

Computer Aided Engineering (CAE) is new technology which is used now days for the fulfilment of the purpose mentioned above i.e. good quality, cost effectiveness, safety, comfort, durability etc. With the help of this tool design engineer can do various type of analysis in short period of time which is relatively easy process as compare too old days where one has to calculate manually and often commit mistakes in complex structures. CAE includes Computational Fluid Dynamics (CFD), Finite Element Analysis (FEA), Multi Body dynamics (MBD) & Optimisation.

The benefits of CAE include:

(i) Saves money and time as designed product can be evaluated various time with different condition using designing based software's earlier these testing was done physically which was relatively very expensive process.

(ii) Solve complex problems quickly.

(iii) Give 3D view and make problem easy for user to understand. Now days teachers are using these software's to give better understanding of concept for their students.

(iv) It helps engineers to predict the performance and cost effectiveness of their new innovative designs.

(v) Help in increasing product life & decreasing cost when integrated with manufacturing as it will predict any problem related to manufacturing and its resolution before production of product.

(vi) Graphical presentation of results with great celerity.

1.2 Some Important software used by Design Engineers

The different CAE software used by design engineers are:

I. ANSYS
II. ABACUS
III. Hypermesh
IV. PATRAN
V. DYNA
VI. I-DEAS
VII. COMSOL
VIII. Pro-Engineering
IX. CATIA

1.3 Methods of solving any engineering problem

There are three methods to solve any engineering problem.

1. Analytical Method
2. Numerical Method
3. Experimental Method

1.3.1 Analytical Method

It is classical approach in this traditional mathematical formula are used to find the unknown at desired location of body under analysis it uses formulas & equations. It gives accurate results and is best for simple problems like stress, strain, deflection of simple beam, cantilever under simple forces and constraints. It takes more time as compare to numerical method

1.3.2 Numerical method

When it comes to solve any complex problem i.e. any problem having complex forces, constraints we were not able to get solution by analytical method so we started using numerical method it enables the engineer to solve practical design problems by taking some approximations if required to get acceptable solution.

1.3.3 Experimental Method

In this value of unknown is found by physical measurement with the help of various measuring instruments. Thus it gives highly accurate results the drawback of this method is that it is expensive and time consuming as we need to have full set up for performing experiment on the work piece which we need to analyse.

Table 1.1 Comparison of three methods of solving engineering problem.

Sr. No	Analytical Method	Numerical Method	Experimental Method
1.	Classic approach	Mathematical representation	Actual measurement
2.	Nearly Accurate result	Approximate results	100% Accurate results
3.	Mathematical formulas and equations are used	CAD model is needed	Physical body or part on which experiment can be performes
4.	Mostly applied on simple problem	Applied on complicated Problems	Applied on physical problem
5.	Results are dependent on mathematical formulas & equations	Results are not exact they have some deviations as there are some approximations considered.	Results are exact and near to real life solution but it is recommended to take various reading before final conclusion

1.4 Introduction to Finite Element Analysis

FEA is technique used for obtaining approximate solutions to a variety of simple and complex engineering problems. It has received attention of various industries as an analysis tool because of its flexibility. Now days we see that in practical engineering problems where we need to find approximate solutions rather than exact closed form solution. It is used in designing of new product and refinement of existing product. Basic law is to calculate only limited (Finite) number of elements made up of nodes and then integrate the results for entire (volume or surface).

The first step in FEA is to do meshing that is to divide body surface or volume in to number of small elements on which analysis is performed in this way a body having infinite degree of freedom reduces to finite degree of freedom.

Elements are made up by various type of combination of nodes e.g. quadrilateral, triangular, tetras etc. To get value of variable (say stress) anywhere in between the nodes calculations point interpolation function as per the shape of element is used to evaluate results.

FEA is numerical based method and it has wide range of application from total deformation, stress analysis etc. of ships, automotive, aircraft, bridge and building structures to field analysis of fluid flow, heat flux, magnetic flux and other flow problems with development of various designing software complex geometry can be made easily and analysed with great clarity and precision because of this character it play vital role in key industries such as aerospace, defence, Medical, Oil & Gas, automotive, consumer product etc.

FEA is a three stage process that are Pre-processing, Solving and Post-processing.

I. PRE-PROCESSING

Pre-processing is the initial step in this complex geometries are divided in to small simple shapes (elements) and this process is known as meshing. Forces and constraints are applied on part and after that the part model is written in that language which solver can understand for its processing.

II. SOLVER

Now days there are various commercial software's in market to perform Finite Element Analysis some of them are Hypermesh, Ansys, Ls-Dyna, Abaqus etc. They calculate stress, strain, deformation, frequencies etc. These software got some algebraic and numerical equations in matrix form which make it easy for them to calculate as they got input of constraint and forces the user gives solving command the software will check that all input conditions are right if it founds all right then solution computation will start and according to input condition analysis is carried out on each element and output get generated at various points.

III. POST-PROCESSING

After solving the given problem now it comes to its visualisation of solved data this part contain 3D quality colour contoured plots and animation showing results where user can visualise result and can see if any change is required in intial step for getting results.

1.5 Different Types of Analysis Using CAE

The different types of analysis that can be done using CAE tools are

1. Static Structural Analysis
2. Dynamic Analysis
3. Buckling analysis
4. Thermal analysis
5. Fatigue analysis
6. CFD analysis
7. Crash analysis

1.5.1 Static Structural Analysis

It determines the influence and effect of steady load condition on body. It ignores damping and inertia that came due to time- varying loads. But, it includes steady inertia loads *i.e.*, rotational velocity and gravity. There are various type of loads applied during static structural analysis this include external force and pressure, effect of temperature, gravity & rotational velocity having steady state inertial force.

Static Structural Analysis can be classified in to two type depending on the loading, type of material, forces etc.

I. Linear static structural analysis

In this type of analysis there is linear relationship between the two variables .It represent one of the basic analysis. If we apply load on a beam at an end and keep varying that load then we find that the displacement of that end on which load is applied start varying as load varies that if load increase then displacement will increase and if load decreases then displacement will decrease this type of condition have linear relationship so we can say that displacement and load are linearly proportional. Hooks law comes under this category. Stress is directly proportion to Strain.

Linear static structural analysis follows equation (1.1).

$$[K]\{u\} = \{f\} \qquad (1.1)$$

Where, [K] is the stiffness matrix of a system,

{u} is the displacement vector

{f} is the applied forces vector. Using equation (1.1), stresses, strains and other secondary variables are calculated.

II. Non-linear static structural analysis

When the relation is not linear between two variables or if applied force is not directly proportion to the response that type of analysis is known as Non-linear static structure analysis .The nonlinear behaviour occur mainly due to material property of body under analysis, geometry of body, boundary conditions & complex forces applied on body.

In case of material when stress is not directly proportion to strain the material may be elastoplastic in nature.

In case of geometry there may be some geometrical deformations on the body and due to which there may be large difference in stress, strain, stiffness in part of same body

When it come to the boundary conditions and forces they also play role in non-linear static structure analysis if analysis is going on and in between analysis we change the boundary condition or forces then we will get non-linear response.

1.5.2 Dynamic Analysis

If load is varying with respect to time then the response of structure will vary with time so we cannot rely on the result given by static analysis we have to go for dynamic analysis for response of body which is varying with time. It has practical applications e.g force on brackets of automotive running or bumpy road etc. Popular software used for dynamic analysis are Nastran, Ansys, Radioss, Abaqus, I-deas NX, Matlab etc.

1.5.3 Linear Buckling Analysis

The buckling occurs in body due to compressive loads .Linear buckling analysis deals with compressive load only .It use to analyse the sheet metal parts ,beams etc. Critical value of load is output of analysis. This analysis commonly used by civil engineers during construction of multi-story buildings. Mechanical engineering applications- vacuum vessel, long gear shifted rod analysis etc.

Popular software used for linear buckling analysis are Anys, Nastran and Abaqus etc.

1.5.4 Thermal Analysis

Temperature play a wide role in stability of structure so thermal analysis is use to predict the response of structure or body to get a successful design there must be knowledge of thermal flux, Structural response to thermal gradient. Its applications are wide it is used in designing of Engine, exhaust system, radiator, power plants, heat exchanger satellite design etc. popularly used softwares are Ansys, Nastran, Abaqus, I-deas NX etc.

1.5.5 Fatigue Analysis

When subjective to repetitive load there induce fatigue in the object fatigue analysis is used to calculate life of structure ε-N (alternating strain vs. reversals) or S-N curve (alternating stress vs. cycles) is the base for fatigue calculation (like σ-ε diagram for static analysis) .due to fatigue there is 90% failure in structure it occurs in body which are on continuous dynamic loading e.g. automobiles, lathe, milling, etc. popularly used software: MSC Fatigue, FEMFAT, FE SAFE, LMS etc.

1.5.6 CFD Analysis

Computational fluid dynamics as the name suggest it is branch of science which uses numerical and algebraic equations for dynamic analysis of fluid It is based on the Navier- Stroke equations (Mass, Momentum and Energy conservation equilibrium equations). It has wide application in drag prediction and stream lining of an Aircraft, design of combustion chamber to check an optimum fuel – air mixing, car design etc. popularly used Software**:** Fluent, CFD Expert, Star CD, CFX, etc.

1.5.7 Crash Analysis

Crash test are performed to see what deformations happen after crash, where more stress & strain occurs in the body how much a body can absorb the shock which was originated due to collusion with stationary and moving body so that necessary changes in design can be done. Car companies uses crash analysis test to improve safety of passengers. Mainly used software's are Madymo, LS-Dyna, Radioss, Pamcarsh, Abaqus-Explicit, etc.

1.6 OPTIMISATION

We can define optimisation as within the chosen concept and within the constraints on function, make the product as good as possible. For a car it would be natural to minimize cost, perhaps indirectly by using the least possible amount of material, changing shape etc. Optimisation is to get the best result under some provided constraints and circumstances, to find out the optimized distribution of material to satisfy the provided constraints.

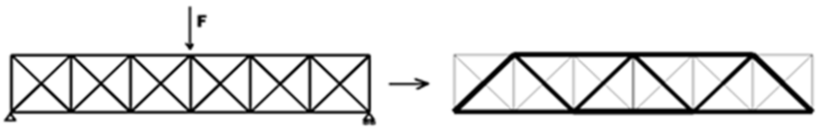
The objective for the problem of optimization consists of constraints limits and parameters that are to be minimized or maximized. The optimal solution obtained depends upon some design variables constraints, forces that can be expressed with numerical value.

The mathematical design optimization method is conceptually different from the iterative-intuitive one. In this method a mathematical optimization problem is formulated,

Where requirements due to the function act as constraints and the concept "as good as possible" is given precise mathematical form.

1.6.1 Type of Optimisation Method

(a) Shape Optimization



(b) Sizing Optimization

(c) Topology Optimization

Figure 1.1.1 Different type of optimization methods [1]

a) Shape optimization

It is a part of optimal control theory. In shape optimization specific design parameters are considered and are varied until constraint and design responses are fulfilled. In its algorithm removal of holes etc. is not allowed it only changes radius, material thickness, chamfer, fillet etc.

b) Size optimization

Size optimization defines ideal parameter such as cross-section dimensions, thickness material values It is used to determine the material ideal thickness that is based on the performance and expected forces that is to be placed on the component during its life. It is performed after free form optimization i.e., when the initial geometry of the component has been defined and interpreted **[24]**

c) Topology optimization

It is a mathematical approach that optimizes layout within a given design space, for a given set of boundary condition and loads such that the final layout meets a desired set of targets. It is a powerful technique which increase creativity both at micro and macro level .It is implemented on material layout through the use of FEM the technique is based on level sets, moving asymptotes and genetic algorithm. It helps in reducing the design development time and overall cost while improving design performance. In some cases result although optimal but may not be feasible for manufacturing then we use manufacturing constraints.

d) Topography optimization

Topography optimization approach is similar to topology optimization approach except that density variables shape variables are used. It is preferred for sheet metal parts and components. In topography optimization density remains same and surface topography is varied to get an optimal solution.

e) Topometry optimization

It is a type of sizing optimization but the difference is that in this technique each element is designed independently not like sizing optimization technique in which all elements that are associated are designed with the same values.

Table 1.2 Different types of optimization methods [2]

Type of Optimization method	Topology	Topography	Topometry	Sizing	Shape	Freeform
Stamping	Yes	Yes	-	Yes	Yes	Yes
Casting	Yes	Yes	Yes	-	Yes	Yes
Extrusion	Yes	**-**	-	-	-	-
Tailor	-	-	Yes	Yes	-	-

1.6.2 Method Used in Shape and Topology Optimization

The aim of shape and topology optimization is to reduce the mass of body and increase the strength. There are some popular methods explained below.

a) Homogenization method

It is a method in which a porous structure was introduced and the optimization was carried out by increasing or decreasing the size of pores. It helps in creating and deleting holes in structure but sometime in complicated cases it gives result that is not manufacturable.

b) Solid Isotropic Microstructure With Penalty (SIMP)

SIMP was developed as an advancement and improvement to homogenization method. This method finds the region of intermediate density and penalizes the region to converge it on a more solid region.

c) Evolutionary Structural Optimization (ESO)

It starts with initial finite model; based on specific objective the elements with least contribution are removed. This method is successful in obtaining local optimized structure.

d) Level Set Method

Osher and Sethian were the first to establish level set function to numerically track moving boundaries [25]. The complicated shape and topology is handle with help of moving zero level set up and down with in the higher order function as shown in figure 1.2

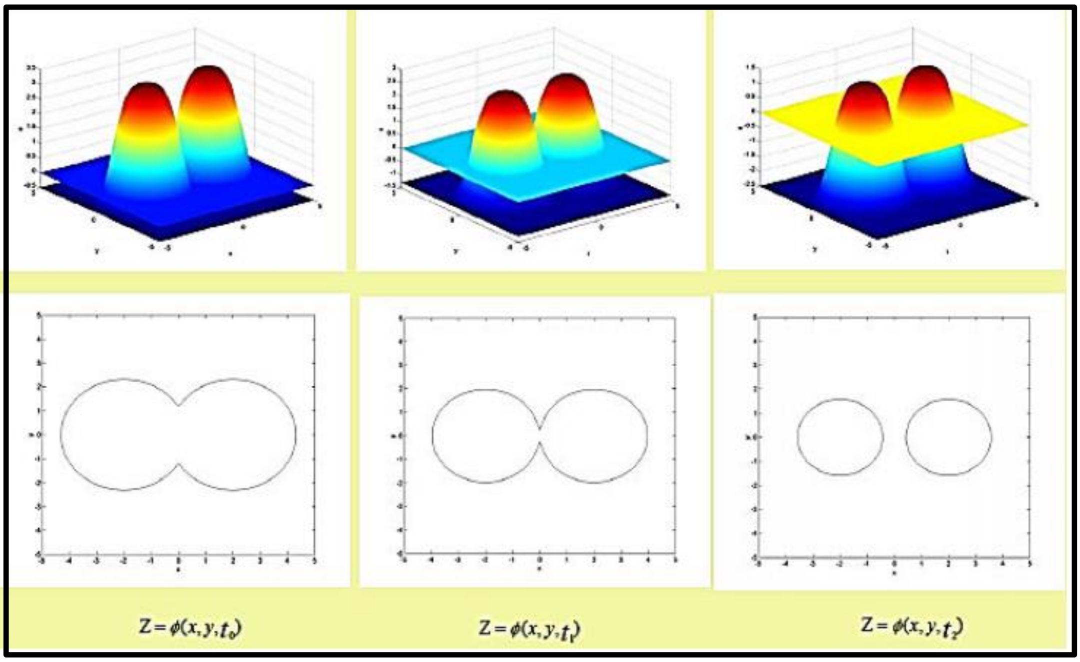


Figure 1.2 Moving the zero level set [26]

1.7 Mathematical background for optimization

The problem is set on an open bounded set, Ω, of real number Dirichlet and Neumann boundary conditions are applied on the boundary ∂Ω.[27]

$$\Omega \subset \mathrm{R}^2 \qquad (1)$$

$$\Omega \ \text{ts}\ \partial\Omega = \Gamma_D \, \mathrm{U} \, \Gamma_N \qquad (2)$$

The displacement field u in Ω is the solution of (3) where A is defined from Hooke's law and the strain tensor is defined by (5)

$$\begin{aligned} -\operatorname{div}(Ae(u)) &= f \text{ in } \Omega, \\ u = 0 & \quad \text{on } \Gamma_D, \\ (Ae(u))\,n = & \quad \text{on } \Gamma_N \end{aligned} \tag{3}$$

$$A\xi = 2\,u\xi + \lambda(\mathrm{Tr}\,\xi)\,\mathrm{Id}, \tag{4}$$

$$e(u) = \tfrac{1}{2}\left(\nabla u + (\nabla u)^t\right) \tag{5}$$

If objective function is compliance with volume and perimeters are constraints, the total amount of work done (in compliance) is.

$$J(\Omega) = \int_\Omega f.u\,dx + \int_{\Gamma N} g.\,u\,ds = \int_\Omega Ae(u)\,.\,e(u)\,dx \tag{6}$$

For a fixed volume, all admissible shapes are defined by $\mathcal{U}ad$:

$$\mathcal{U}ad = \{\Omega \subset D \text{ such that } |\Omega| = V\} \tag{7}$$

The optimization problem, with the addition of volume and perimeter Lagrange constraints, becomes the objective function minimization problem.

$$\operatorname*{Inf}_{\Omega \varepsilon \mathrm{Uad}} \quad J(\Omega) + \ell\,|\Omega| + \ell'\,|\partial\Omega| \tag{8}$$

The shape derivative use to solve optimization problem.

$$J'(\Omega)(\theta) = \int_{\Gamma 0} \left((\ell + \ell' H - Ae(u).\,e(u)\right)\,\theta.n\,ds \tag{9}$$

To track the shape and topology of the structure, the level set function defined as

$$\begin{cases} \psi(x) = 0 \iff x \in \partial\Omega \cap D, \\ \psi(x) < 0 \iff x \in \Omega, \\ \psi(x) > 0 \iff x \in (D \setminus \overline{\Omega}). \end{cases} \tag{10}$$

Is used at any time t, the boundary is represented by the zero level set.

$$\frac{\partial\psi}{\partial t} + \dot{x}(t)\cdot\nabla\psi = \frac{\partial\psi}{\partial t} + Vn\cdot\nabla\psi = 0. \tag{11}$$

By setting the volume equal to: $V = -(\ell + \ell' H - Ae(u)\,.\,e(u))$ (12)

The shape derivative (9) is used to solve (11)

CHAPTER 2 LITERATURE REVIEW

2.1 Literature Review

A number of researchers have worked on the analysis and optimization of different automobile components. The following section gives the review of such work.

Choi *et al* **[3]** carried out a test in which effect of acceleration were measured on the battery fixing bracket of electrically driven automotive in different road conditions (highways, local roads, unpaved road and city street). Single axis and six axis acceleration test methods were used for finding the vibration history on real road conditions and it was noted that the single axis acceleration test method was more reliable and faster than six axis acceleration test method.

Wijaya *et* al **[4]** carried out a test in which an air conditioning system hose used in automobile was considered as a problem. Dynamic analysis were carried out on it i.e. modal and transient analysis using Finite Element Method. The results were validated experimentally.

It was found in first six natural frequencies that the swing and bending modes of the hose occurred. Transient analysis showed the maximum stress occurred in the reinforced braid layer of hose component.

Schramm *et al.***[5]** carried out a test in which structural optimization technique was used for designing of automotive elements and structure. Some manufacturing constraint and distinguish techniques were used to solve the problem of manufacturing of components .The first technique was for topology optimization in which manufacturing constraints were introduced so as to get element or structure easily manufacturable and in second one number of spot welds were reduced in a vehicle with help of topology optimization.

Sheldon Imaoka [6] Provided information about basic difference between two automated method of generating constraint equation. For tying together dissimilar meshes or distributing loads through bolt connection the use of RBE3 and CERIG is recommended. It was concluded that linear constraint equations are used in ANSYS so they are not valid for large complex or rotational-analysis. So beta element, RIGID184 are used, for generation of rigid connection in non-linear problem.

Chang [7] discussed the different methods of optimization used for the mounting brackets to get more stiffness, strength and less weight. The situation under which different optimization techniques to be applied were discussed.

It was concluded that size optimization should be applied when design of a component is to change when it is near to mass production. When the time for mass production of component is more, topology optimization is preferred to change the design of a component. But, if there was no restriction in time limit, both the optimization methods can be applied.

.

Raghavan *et al.* **[8]** In this paper finite element approach was used for analysis and optimization of hydro-mount bracket by reducing its weight. Hydro-mount bracket was used to protect the fluid filled rubberized inner mount from shock loads. Modelling was done with help of solid elements. ETA-VPG pre-processor was used for static analysis. An alternative design was proposed in which weight was reduced. Experimental verification of static performance and durability was done. Finally 8% of weight was reduced after making some minor modifications.

Xiao *et al.* **[9]** carried out a test in which topology optimization was performed on the electric bicycle frame to reduce mass and increase structural stiffness under dynamic conditions Geometric, mechanical and finite element models, as well as a flexible coupling dynamic model was constructed. Through real life testing accuracy and validity of these models were investigated. All key points were extracted by applying dynamic load and road excitation. The modal analysis of original and refined model was performed and structural stiffness was evaluated. The result indicates that optimization was effective to develop a new effective bicycle frame.

Erke Wang *et al* **[10]** Performed analytical and experimental analysis for different mechanical problems which were then simulated using FEA with hexahedral and tetrahedral element. The comparison was done for non-linear analysis, linear analysis and modal analysis. After analysis it was concluded that quadratic tetrahedron elements are good and can be preferred over hexahedral elements.

Laxman and Mohan [11] Performed Structural optimization for analysis during initial stage to get most effective and most efficient design. Optimization was performed on three automotive components of a truck i.e., step bar module, shock reinforcement assembly and transfer case assembly with aim to reduce mass without hindering structural stability and requirements of components that are crashworthiness, stiffness, durability and modal frequencies.

Jeong Woo Chang *et al.* **[12]** Topology optimization of compressor bracket was performed using homogenization method with objective to minimise natural frequency. A new bracket was manufactured based on topology optimisation result and comparison of real bracket and refined bracket was carried out and was concluded that refined bracket would not fail during vibration testing under durability condition.

Tan *et al.* **[13]** carried out a test in which heavy truck propeller shaft bracket was considered for structural optimization. Based on finite element analysis and rigid-flexible coupling analysis a model including bracket, propeller shaft, mount and frame was established. Dynamic stress and dynamic response was taken in to consideration, effect of variation of frequency on structure of bracket were analysed and optimize bracket was manufactured and validated experimentally

Sailto *et al.* **[14]** A new topology optimization method was discussed to optimize configuration and part shape of automotive as only solid elements were used during conventional optimization but in this study solid elements are embedded with shell elements with help of this technique stiffness optimization of various parts including full vehicle model, cylinder model and automotive floor model it was concluded that new method was effective for topology optimization of automotive components made up of steel sheets and have rectangular cross sections.

Kumar and Uppala [15] carried out a test using Finite Element Analysis oil strainer mounting bracket behaviour was studied under dynamic loading conditions. Modal superposition method was applied. Frequency response analysis and modal analysis was performed for different frequencies which are generated from different load condition on engine. The stress along with natural frequency was obtained and was correlated with experimental test results. Several Design modifications are carried out using CAE based on correlation results.

Zhu *et al.* [**19]** carried out a test with the help of Finite Element Method approach dynamic behaviour of bracket which was mounted on engine chassis was analysed. Mode shapes and natural frequency of bracket were evaluated. During analysis of engine mount the influence of bolt holes, C-section edges, boundary constraints and ribs were considered.

Edas *et al.***[20]** With the help of Finite Element Method approach optimization of exhaust design was carried out with aim to reduce the weight and cost. Static and dynamic analysis were carried out to solve the optimization problem gravitational load was considered for analysis of exhaust. The weight reduction was achieved in case of dynamic analysis there was 22% reduction in weight and in case of static analysis 25% reduction of weight.

Fukushima *et al.* **[21]** Homogenization method was used for topology and shape optimization of various car body problems under multiple loading conditions with objective to minimize the design compliance. With the help of topology optimization the problem of strength and validity of car body was solved.

Alzahabi *et al.* **[22]** Optimization of bracket was done by reducing its mass and increasing its stiffness and strength with the help of computer aided design the constraint was used is volume and stiffness was maximised as volume available was less. Dynamic analysis of optimized bracket was done and it was found strength and stiffness of bracket was improved under modal requirements.

Subbiah *et al.* **[28]** Failure analysis of muffler mounting bracket of three wheeler is observed during durability test. The muffler mounting bracket and engine is located on rear end of three wheeler. The problem of crack between engine cradle and crack were observed after 10000km on an average. Using fish bone diagram various possible causes of failure was identified. Statistical analysis was performed using weibull distribution of durability life prediction. After this FEA was used for the solution to the problem the CAD model is made and meshing is done with shell element instead of 3D elements for sheet metal part as structure represent much more efficiently and without compromising with accuracy. The boundary conditions and forces were applied accordingly and stress and strain were predicted and various weak areas were observed and a new design was made by refining present design and it was concluded that new design is much more effective than previous one.

Rao *et al.* [**29]** This paper examine various type of excitation effect the dynamic characterization results which include variation of damping and stiffness by changing the frequency generated by shaker and electrohydraulic excitation of exhaust isolators used in automobile various type of excitation inputs were given which include acquired road data profile, Sine sweep and shaped random. The paper recommends use of time histories. Except at low frequency stiffness and loss factor were nearly similar between electro hydraulic and shaker excitation

2.2 Gaps in Literature

From the literature review it was found that a lot of work has been done in the area of optimization and analysis for the different brackets used in an automobile. But still there is a lot of work to be done in this field. The following are the gaps that were found during literature review.

1) Limited work has been reported on transient response analysis of silencer mounting brackets.

2) Different optimization techniques like shape, size, tomography, topometry and freeform have not been applied on various parts of automobiles.

3) Limited work has been reported on Non-linear static structural analysis to understand the behavior of material and structure under non- linear conditions and its use in automated optimization.

2.3 Objectives of the Present Work

The following objectives were identified after going through the above literature review

1) Linear static structural analysis to be performed on silencer mounting bracket.

2) Modal analysis and Modal frequency response analysis to be performed to analyze the stresses and displacements at corresponding natural frequencies.

3) Topology, topography and shape optimization to be performed with objective of decreasing mass of bracket and increasing stiffness.

4) Original bracket design and optimized bracket design to be compared based on mass, displacements and stresses.

CHAPTER 3 ANALYSIS OF BRACKET

A market survey was conducted and various automotive brackets were analysed for the present work. Finally a silencer mounting bracket of a commercial light truck was chosen for analysis and optimization. The bracket is mounted on the rear end of light truck and is attached with muffler as shown in fig 3.1

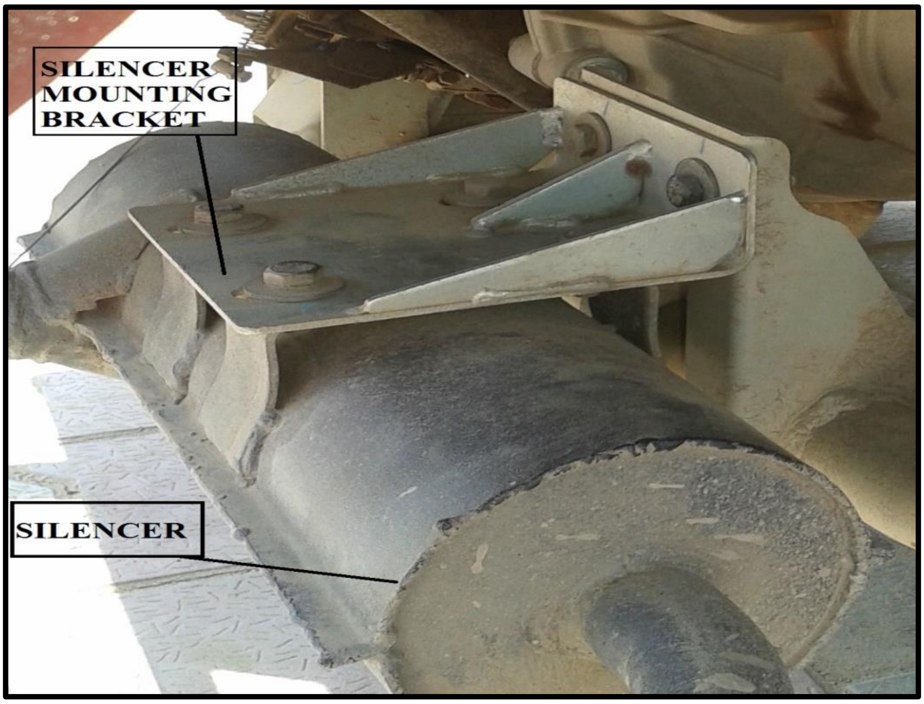


Figure 3.1 Location of silencer and bracket in the vehicle

3.1 Reverse Engineering of the Bracket

As CAD model of bracket was not available so the original bracket was purchased from the market and process of reverse engineering was carried out for making CAD drawing. Various instruments were used for measuring dimension of bracket including Profile Projector, Vernier calliper, screw gauge, depth gauge, Protector, measuring scale etc.

3.2 Material Properties

The composition of material was found with the help of spectroscopy using spectrometer. The percentage composition of iron, silica, carbon, magnesium, phosphorus, sulphur, chromium, molybdenum, nickel, aluminium, cobalt, copper, Niobium, Titanium, vanadium, lead was noted and it was found that material was basic structural steel and further classified as High-strength low alloy (HSLA) steel. The property of table 3.1 was taken from ANS **[16]** for HSLA and shown in Table 3.1

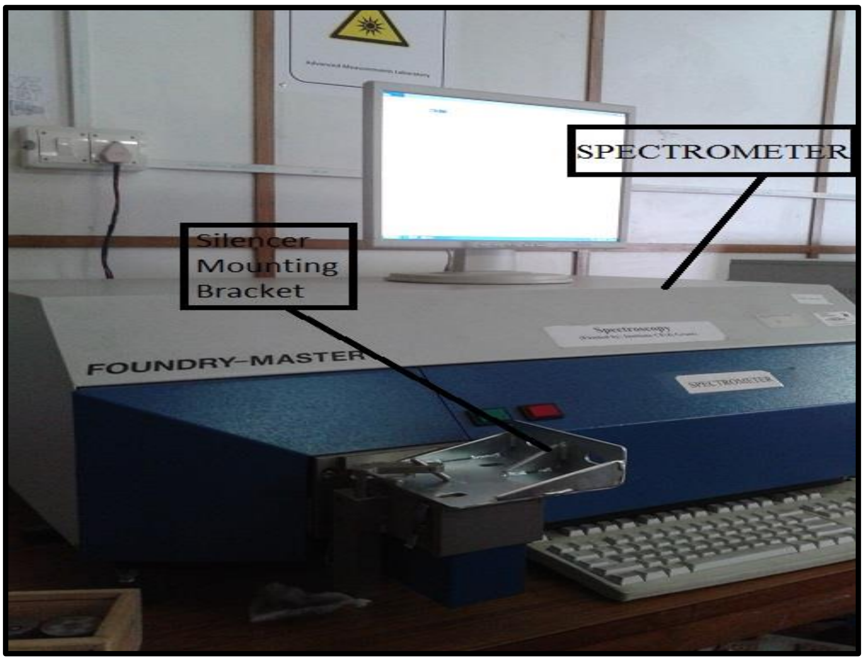


Figure 3.2 Spectrometer Used for Spectroscopy of silencer mounting bracket

Table 3.1 Properties of the Material

Property	**Value**
Elastic Modulus	3.1 e+011 N/m^2
Poisson's Ratio	0.30
Density	7850 kg/m^3
Tensile Yield Strength	300 MPa
Tensile Ultimate Strength	460 MPa

3.3 Modelling and Mesh Generation of Silencer Mounting Bracket

With the help of dimensions which are measured using various instruments a CAD model was created in Pro-e modelling software. The model is shown in Figure 3.3

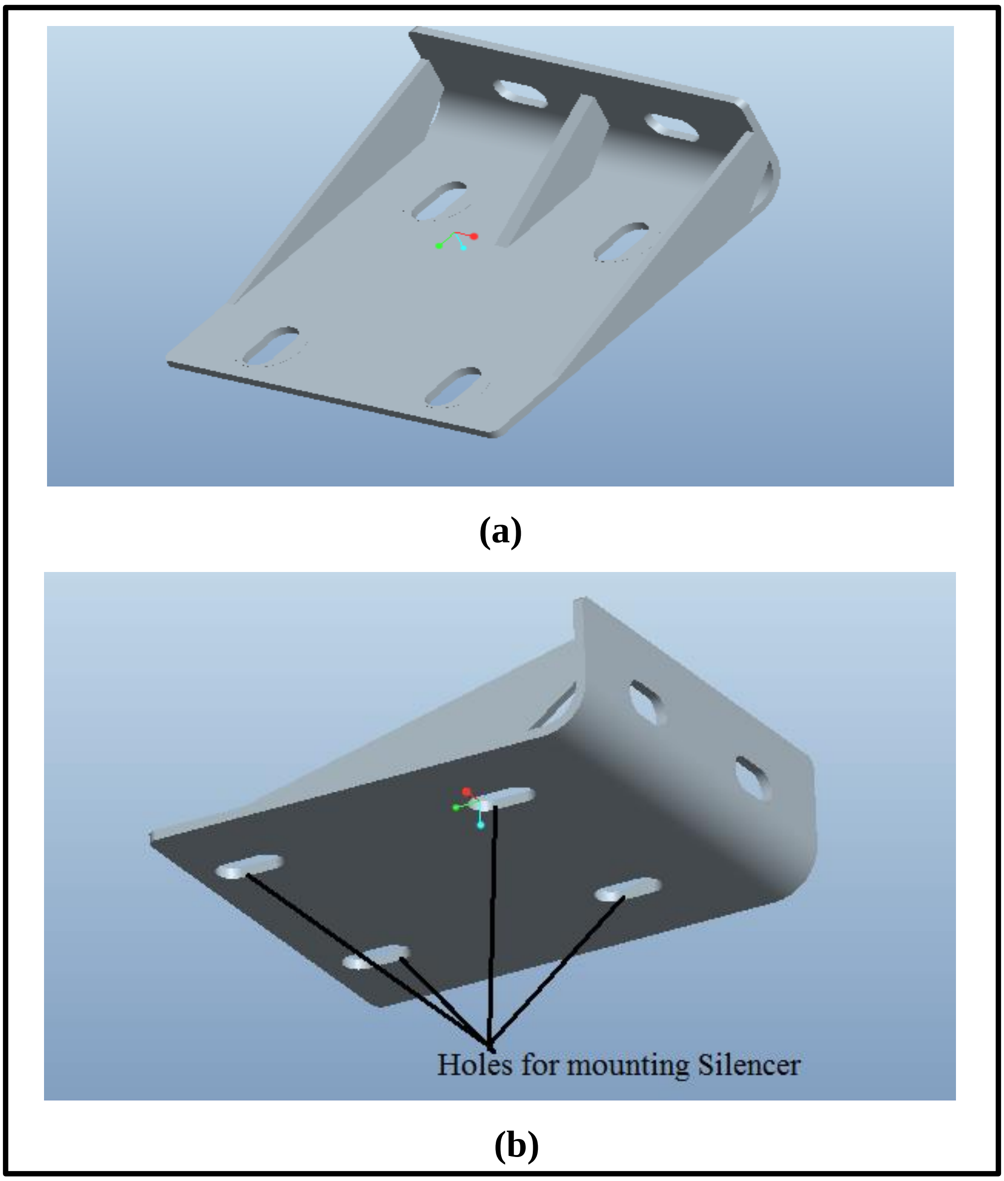


Figure 3.3 a) Isometric view of silencer mounting bracket model, (b) CAD model showing holes for mounting of silencer

3.4 Static Structural Analysis of original Bracket

Static structural analysis was performed by applying load equivalent to weight of silencer and analysis was used to obtain von-mises stress, deformation, Strain etc. Linear static structural analysis was performed using ANSYS 15. After applying boundary conditions and forces on meshed model of bracket the analysis was carried out in ANSYS software.

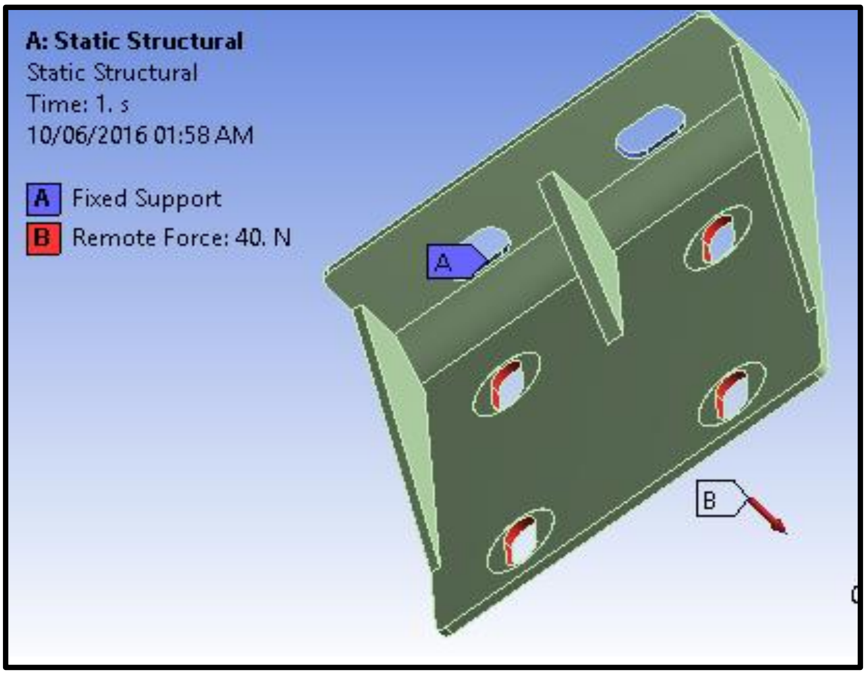
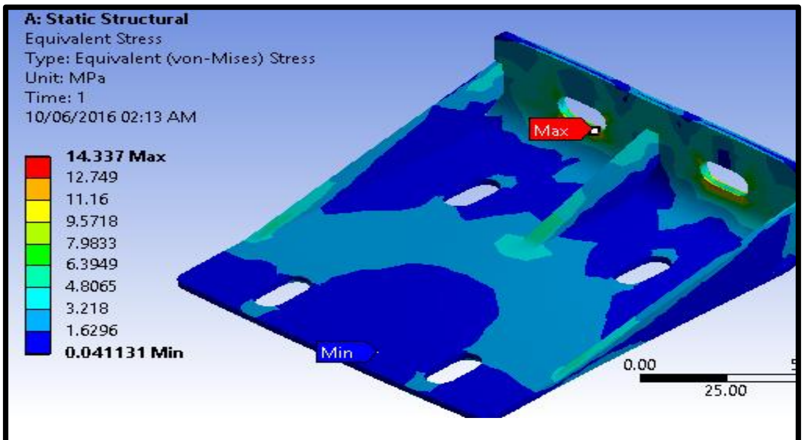
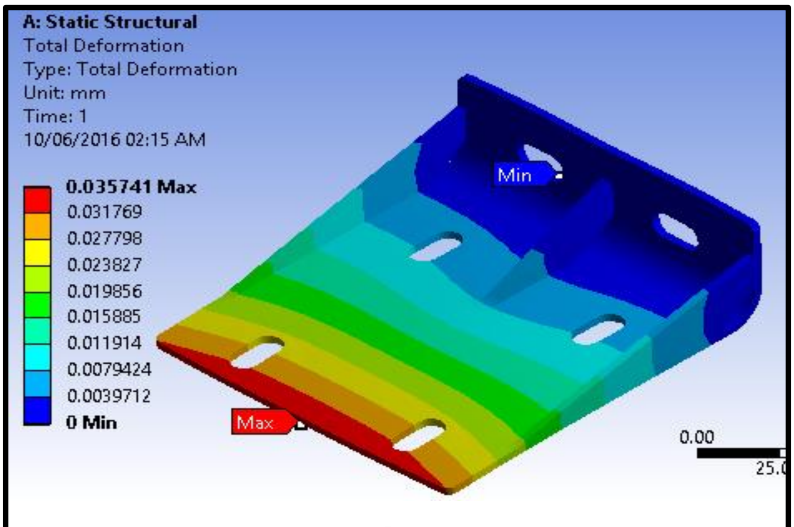


Figure 3.4 Fixed support and Remote force applied for static structural analysis

The result of total deformation and von-mises stress was obtained after static structural analysis of silencer mounting bracket with above force and boundary conditions.



(a)



(b)

Figure 3.5 Results obtained after performing the static structural analysis: (a) Von mises stress (in MPa) (b) Total Deformation (in mm);

It can be seen from figure 3.5 that the bracket has a high factor of safety and is over designed.

3.5 Dynamic Analysis

Dynamic analysis was performed to check the effect of vibration on this bracket. The following procedure was followed for dynamic analysis of bracket:

(i) Modal analysis was performed

(ii) Modal frequency response analysis was done using natural frequency obtained from modal analysis.

The output of modal analysis works as input for modal frequency response analysis.

3.5.1 Modal analysis

Natural frequencies and their corresponding mode shapes are obtained with the help of modal analysis. This acts as a first steps towards dynamic analysis of the part and its results are used as input for random response analysis, frequency response analysis and transient analysis.

The following parameters were taken into consideration for the Modal analysis.

a) **Damping:** for finding natural frequency damping was neglected.

b) **External force:** as per the standard to find natural frequency no external forces were applied.

c) **Output of Analysis:** Magnitude of frequency and mode shapes were the results obtained from this analysis

ANSYS 15 software was used for the modal analysis of the bracket.

The load and boundary conditions are applied as shown in Figure 3.6. The load of 4kg was applied at the centre of mass of the silencer which is mounted downside of the bracket.

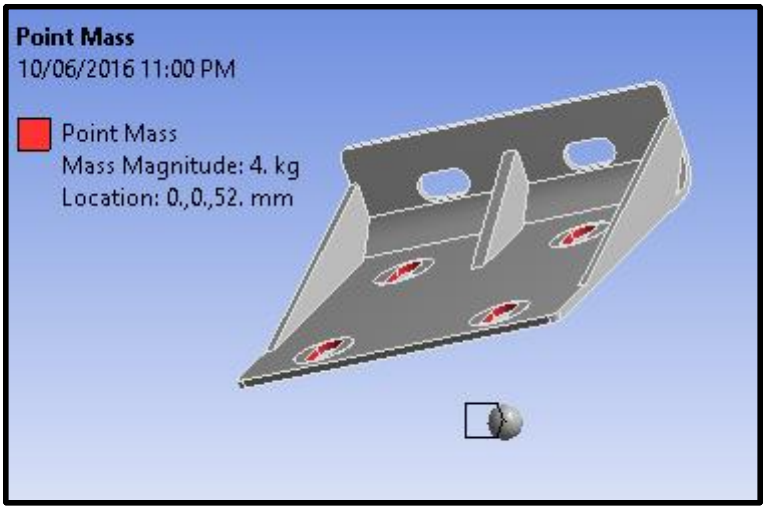
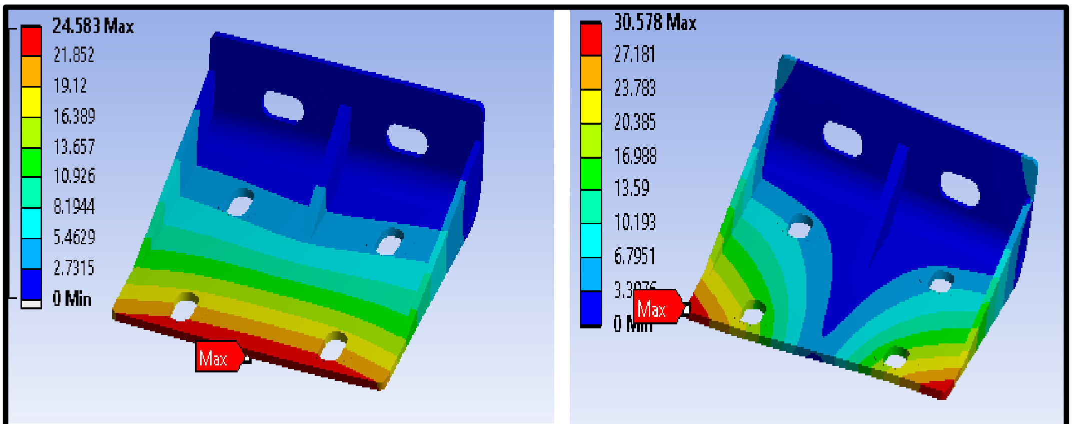


Figure 3.6 Boundary conditions for modal analysis and frequency response analysis

The modal analysis was carried out to find natural frequency and the corresponding mode shapes. The different mode shapes corresponding to the natural frequency are shown in figure3.7.



(a) (b)

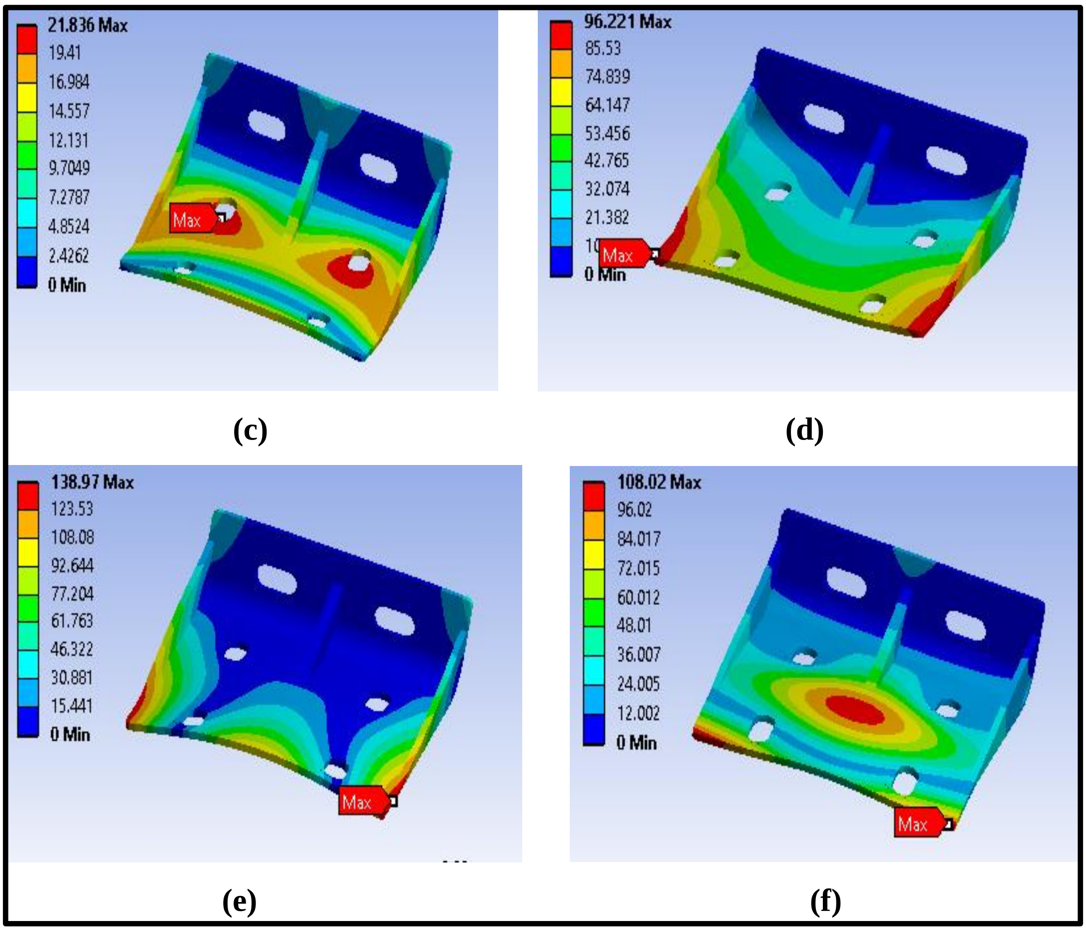


Figure 3.7 Mode Shapes for the bracket: (a) Mode shape 1 at 78.83 Hz; (b) Mode shape 2 at 132.85 Hz; (c) Mode shape 3 at 397.07 Hz; (d) Mode shape 4 at 897.33 Hz; (e) Mode shape 5 at 1501.1; (f) Mode shape 6 at 2035.9Hz

3.5.2 Modal Frequency Response Analysis of Silencer Mounting Bracket

In this analysis external harmonic excitation was given with the help of external force. In this case silencer mounting bracket was excited with harmonic response in the form of $F_0\sin\omega t$, where F_0 is amplitude and ω is frequency of excitation. The high stress and vibrations are caused at time of resonance frequency and it occurs when ω is equal to the natural frequency. The Modal frequency response analysis of the bracket is performed along x, y, and z direction.

3.5.2.1 Modal frequency response analysis along x-axis

As per industrial standards (JIS1601D) **[18]** excitation force was applied along the x-direction. For the analysis of bracket load and boundary conditions was considered as shown in Figure3.8. 40g of inertial acceleration was applied in x-direction which was measured with the help of accelerometer and the damping ratio (actual damping coefficient to the critical damping coefficient) is considered to be 6% as per standards. Figure 3.8 shows the applied acceleration in x-direction.

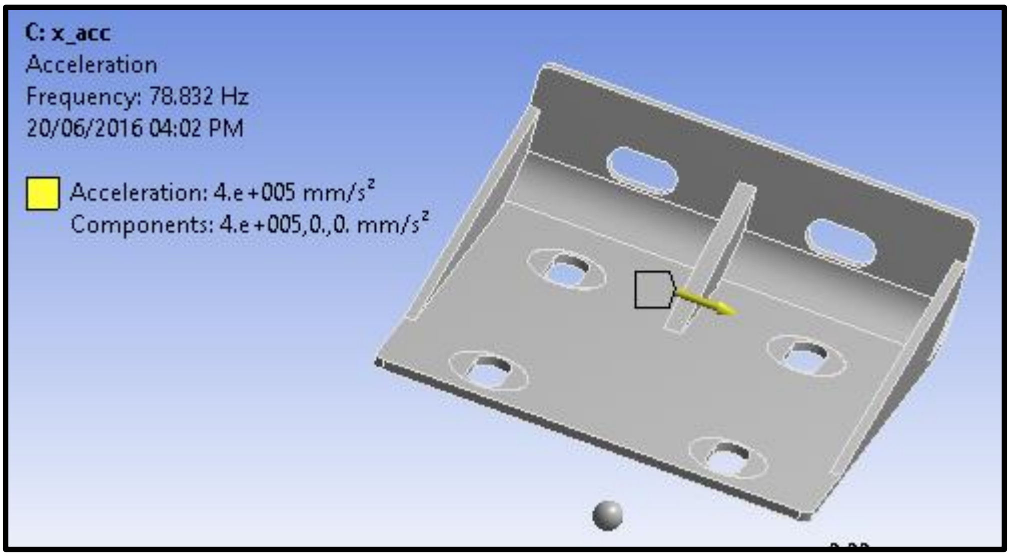


Figure 3.8 Direction of acceleration applied in x axis.

Frequency range was considered to be from 70 Hz to 2400 Hz with 100 numbers of intervals.

Von-mises stress was calculated along x-direction with frequency 78.83 Hz and it was found that stress is under yield strength limit of material Figure 3.9 shows the von-mises stress under dynamic condition.

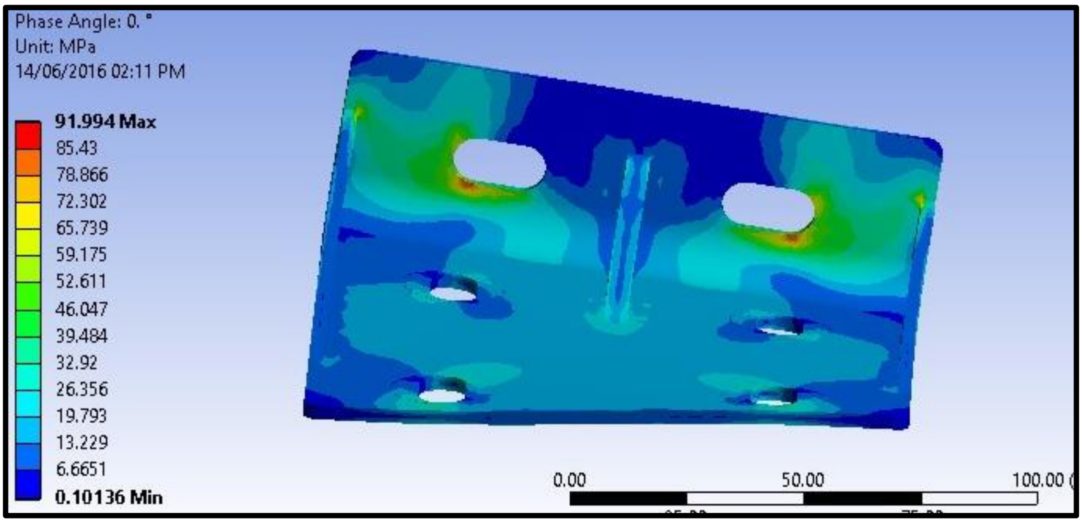
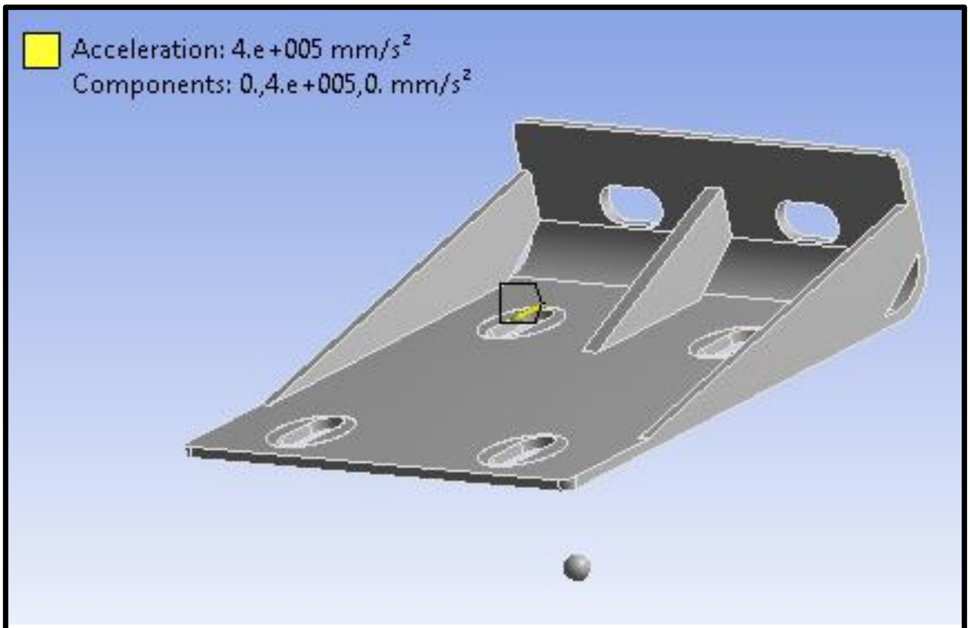


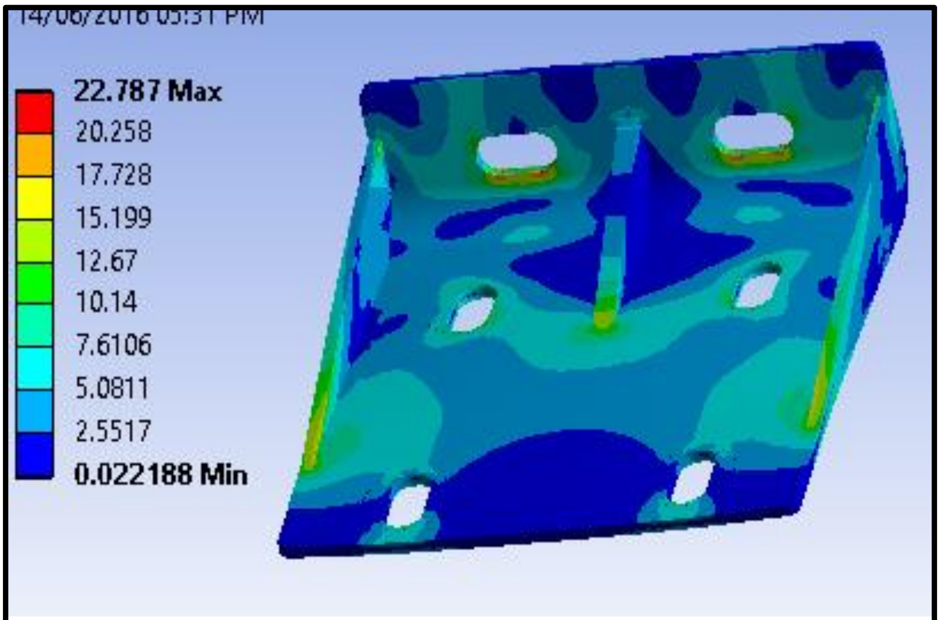
Figure 3.9 Von-mises stresses produced in (x-axis direction) at frequency 78.83 Hz

3.5.2.2 Modal frequency response analysis along y-axis

With 40g of inertial acceleration in x-direction and damping ratio considered to be 6% per standard, von-mises stress was found at the frequency of 78.83 Hz and it was observed to be under the yield strength limit of the material.



(a)



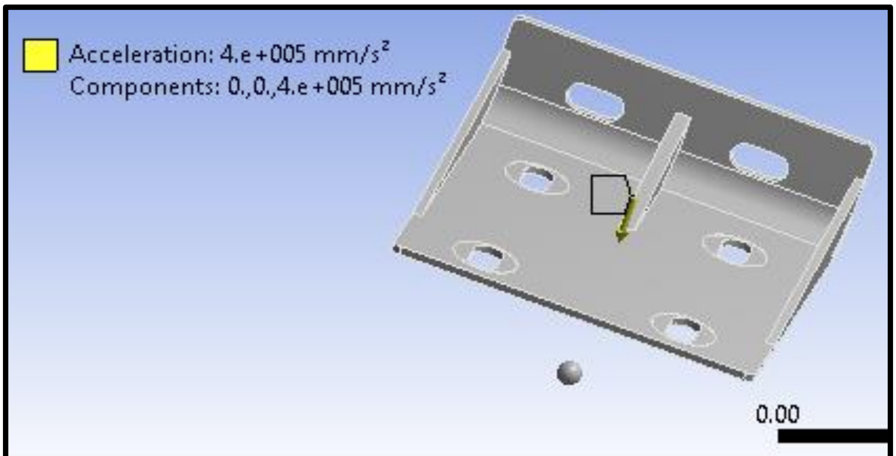
(b)

Figure 3.10 (a) acceleration load applied along y-axis,

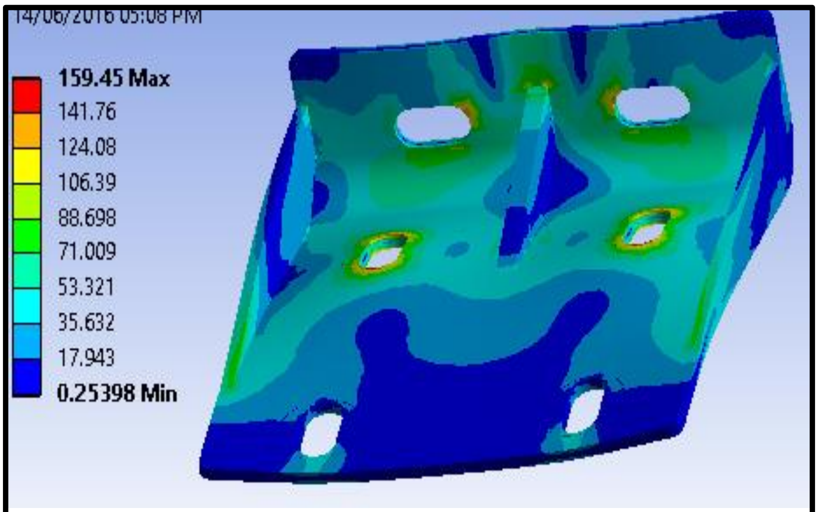
(b) Von mises stresses produced in y-axis at frequency 78.83 Hz.

3.5.2.3 Modal frequency response analysis along z-axis

With 40g of inertial acceleration in z-direction and damping ratio was considered to be 6% per standard, von-mises stress was calculated at the frequency of 78.83 Hz and it was found under the yield strength limit of material.



(a)



(b)

:

Figure 3.11(a) acceleration load applied along z-axis, (b) Von mises stresses produced in z-axis at frequency 78.83 Hz.

CHAPTER 4 OPTIMIZATION OF BRACKET

4.1 Optimization of original bracket

Optimization is a mathematical approach that is used to optimize the given problem under some limits, constraints and forces defined by user. It gives result by considering all the constraints and boundary conditions

The following steps were used for the topology optimization of the bracket.

Base design's shape and mass plays a vital role in final result of topology optimization. Here base design model was made by measuring various dimensions of bracket with help of vernier caliper, Screw gauge, profile projector etc. The model of silencer mounting bracket was made in Pro-e and then saved as in IGES format.

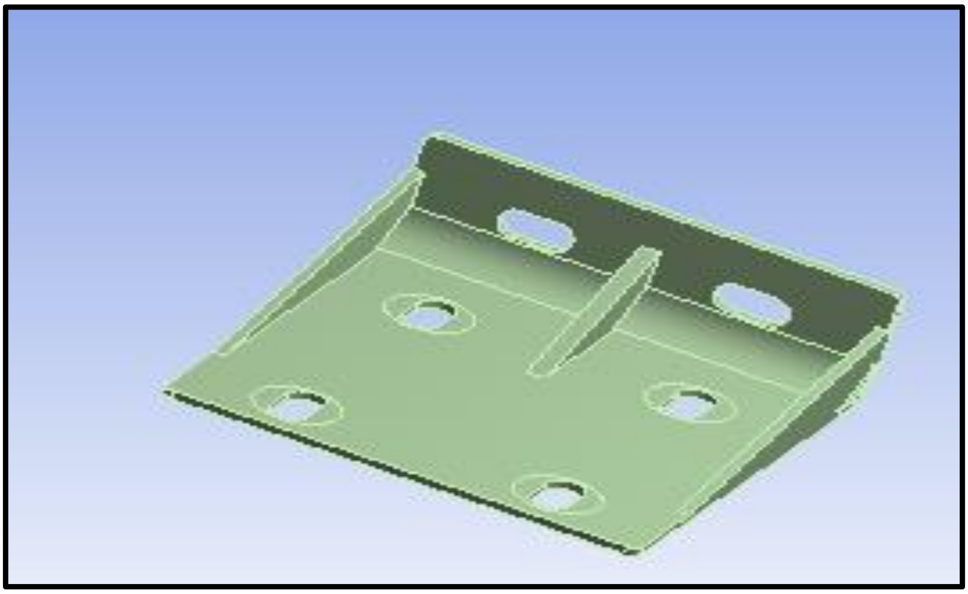


Figure 4.1 View of Base Design of Bracket for Topology Optimization

4.2 Meshing Of CAD Model

First of all design and non-design spaces were defined and then meshing of model was done. There are various type of meshing basically divided in to 1D, 2D and 3D. Out of these best results come using 3D element. Different elements can be used for 3D meshing that are Tetra, Pent or wedge, Hexa or brick and Pyramid. Here in this case tetramesh was used for best results.

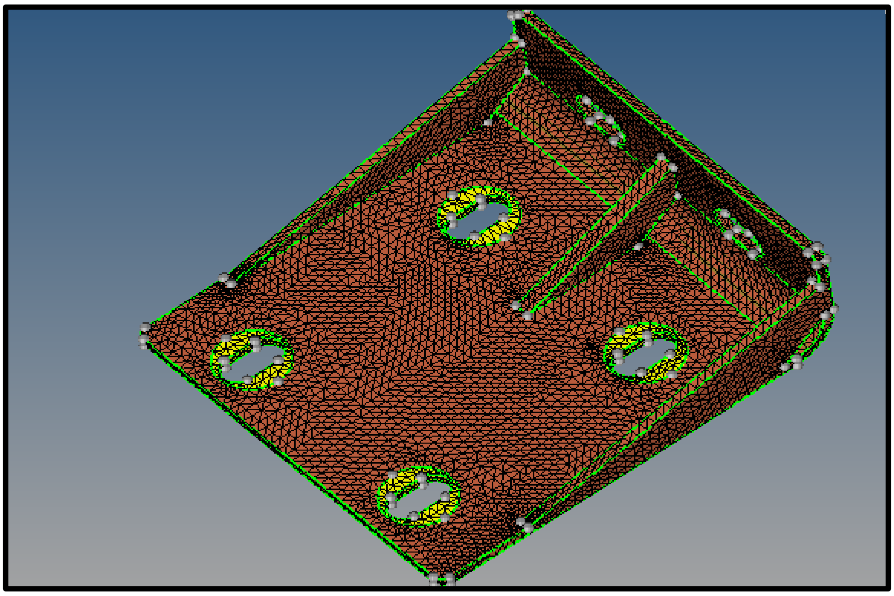


Figure 4.2 Meshed Bracket

4.3 MATERIAL OF BRACKET

Material of bracket is HSLA (High Strength Low Alloy) which was found after spectroscopy. In hypermesh material was created by specifying its various properties.

a) Type as Isotropic and card image as MAT1.

b) Give E, NU and RHO valuse as specified in tabel 3.1

E = Young's Modulus

ν [NU] = Poisson's Ratio

ρ [RHO] = Density.

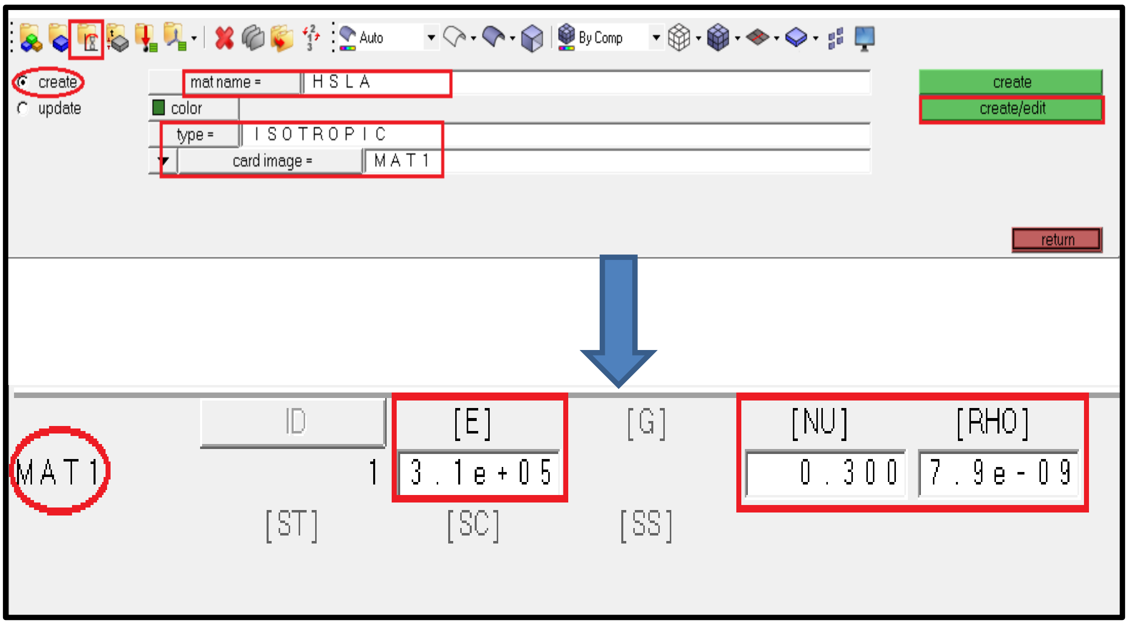


Figure 4.3 Slection of material and its properties

4.4 Creation of RBE3 and RBE2

For Topology, Topography optimization the concept of Rigid Body Element (RBE) is use to define the master and slave nodes on which mass, force, acceleration etc. is acting. In this case we have used RBE3 & RBE2 both.

RBE2: This component was utilized to characterize bolt area. Rigid region was created with the help of related nodes and equations of constrain was generated. It was used to define degree of freedom at dependent node. Two types of nodes are required by RBE2 that are master and slave nodes.

The RBE2 is also known as rigids in this case we use rigids where we had to fix bracket with the body of vehicle

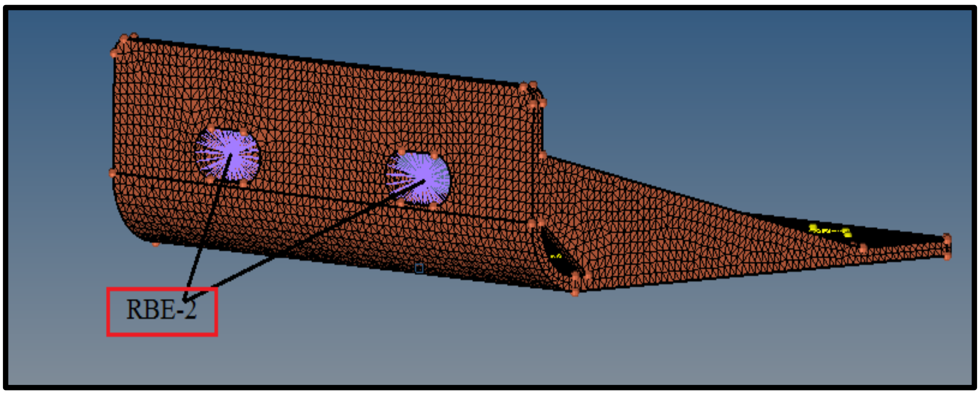


Figure 4.4 RBE 2 element in bracket

RBE3: A rigid region is created by the elements. In this element, slave nodes are independent nodes and master node is a dependent node. Here, in this problem it represents the position of centre of mass of silencer.

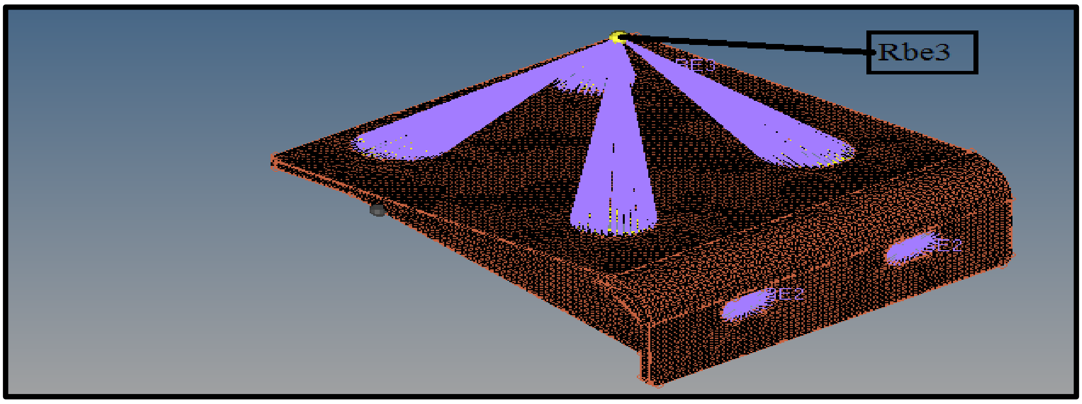


Figure 4.5 RBE 3 element with master node at centre of mass of silencer

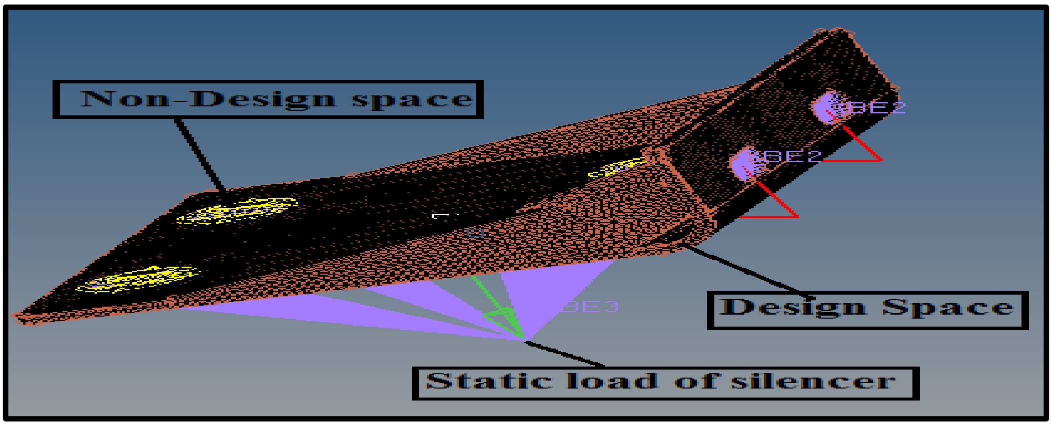


Figure 4.6 Bracket showing design, non-design space, RBE2 & RBE3

4.5 Topology Optimization on basis of linear Static Structural Analysis

After defining material, forces and constraints topology optimization was performed on basis of static analysis of silencer mounting bracket. It is mathematical approach that optimizes given material layout for a set of boundary conditions and loads with in given design space. The load, constraints, design space and non-design space was already defined in previous sections.

4.5.1 Creation of Design Variables

Design variables are parameters that are used to give practical optimized result which is easy to manufacture and save cost and time.

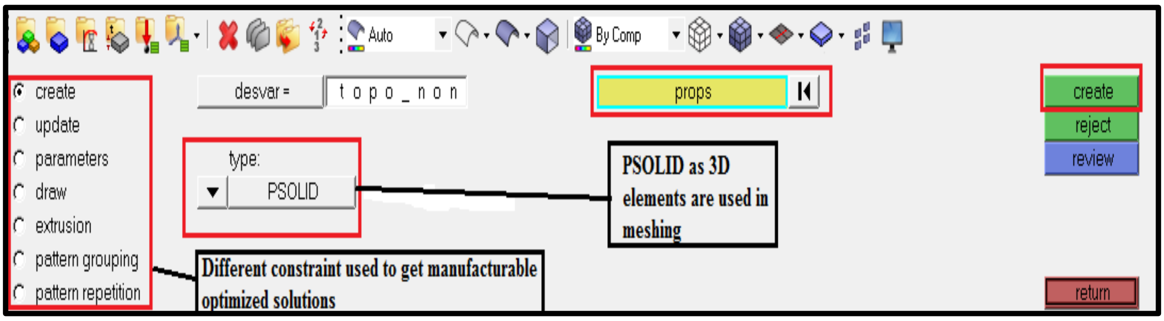
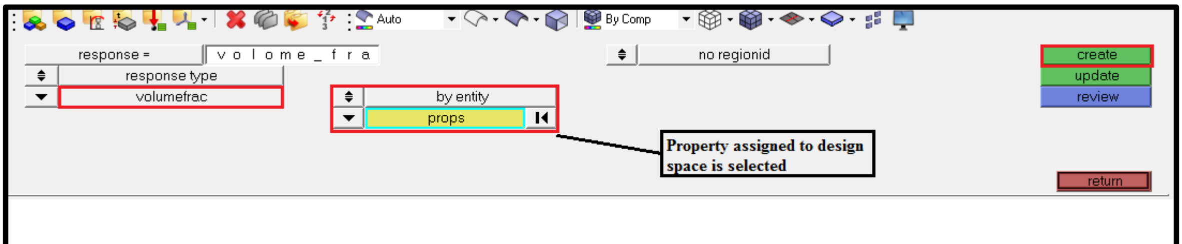


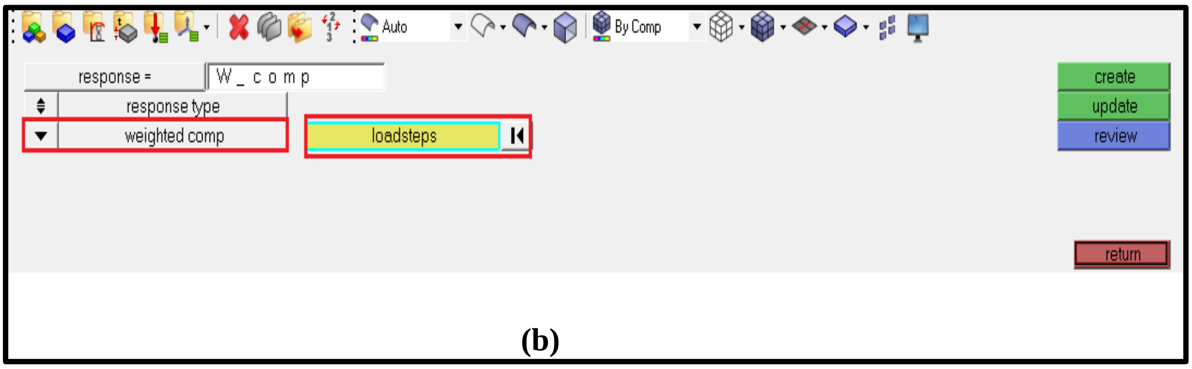
Figure 4.7 Creation of Design variable

4.5.2 Creation of Response

Response is measure of system performance. Two responses were created that is weighted compliance and Volume fraction.



(a)



(b)

Figure 4.8 Two response type defined for topology optimization (a)Volume fraction, (b) Weighted compliance.

Volume fraction response type is applied on design space and its property was defined in section 4.3. Weighted compliance response type is defined with static load step.

4.5.3 Creation of Constraints

Constraint function depends upon response function in this section we give upper and lower bond to response in this case we will give lower bond value 0.750. Response type volume fraction which means that 75% present volume of design space will be used to get optimized layout of silencer mounting bracket.

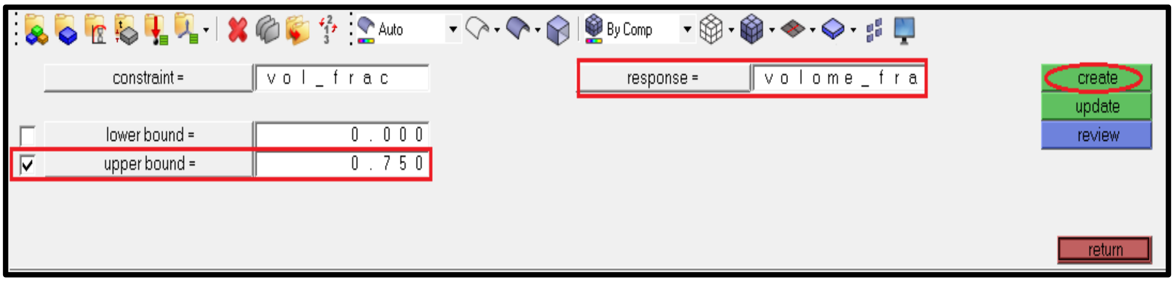


Figure 4.9 Creation of Design constraint4

4.5.4 Defining objective

Objective is response function of system that needs to be optimized. In this problem the objective is to minimize the weighted compliance as weighted compliance decreases the stiffness will increase so it means we get stiffest structure using 75% of volume.

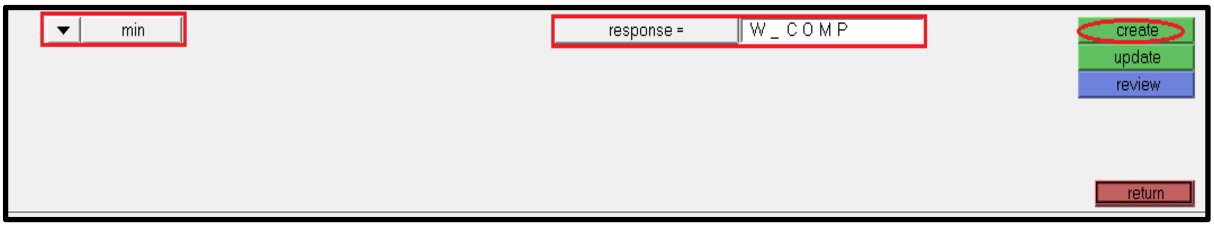


Figure 4.10 Creating objective.

Figure 4.11 of model browser after completing all necessary step for topology optimization with help of static analysis.

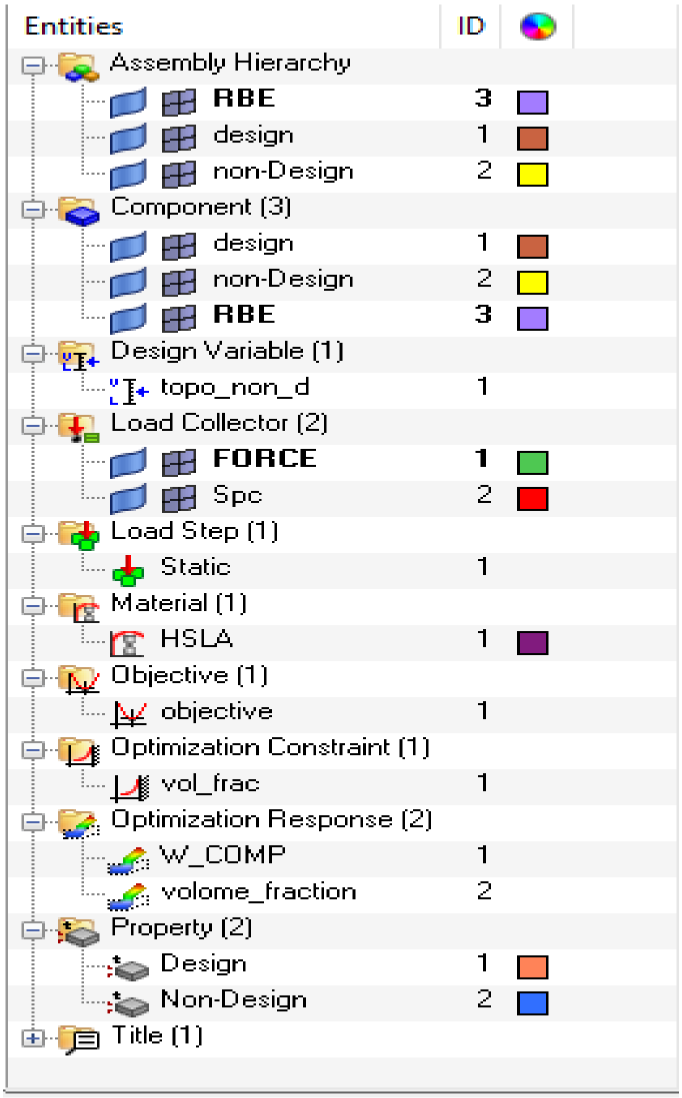


Figure 4.11 Model Browser of Topology optimization on basis of static structural analysis

4.5.5 Result of Topology optimization (Post processing)

Density information was provided by optistruct for all iterations. It also gives information about von-mises stress and displacement on silencer bracket under linear static analysis. To see the result click on hyperview a new window will appear and select last iteration by clicking on iso value toolbar. In this case last iteration was 25^{th}.

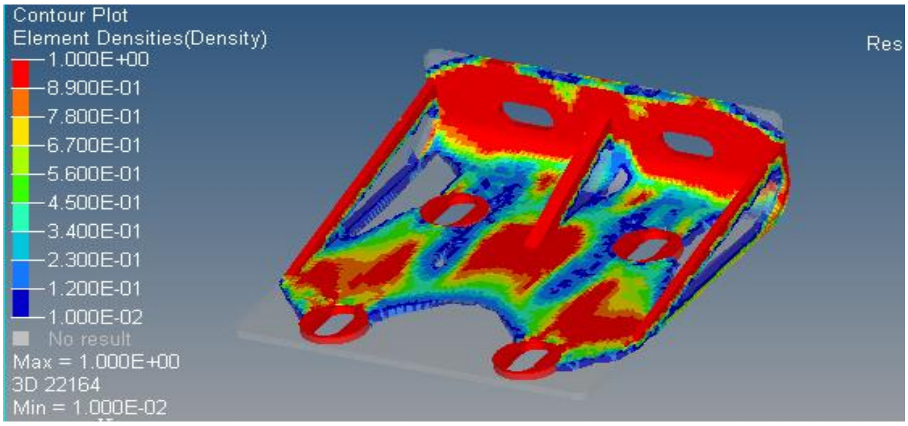


Figure 4.12 Final layout of Topology optimization on basis of static structural analysis

4.6 Topology Optimization on basis of modal analysis.

For topology optimization of material lay out on basis of modal analysis was same as in above case but in this analysis force of 4kg was not applied on master node which was made using RBE3 but mass of 4kg was used on that master node.

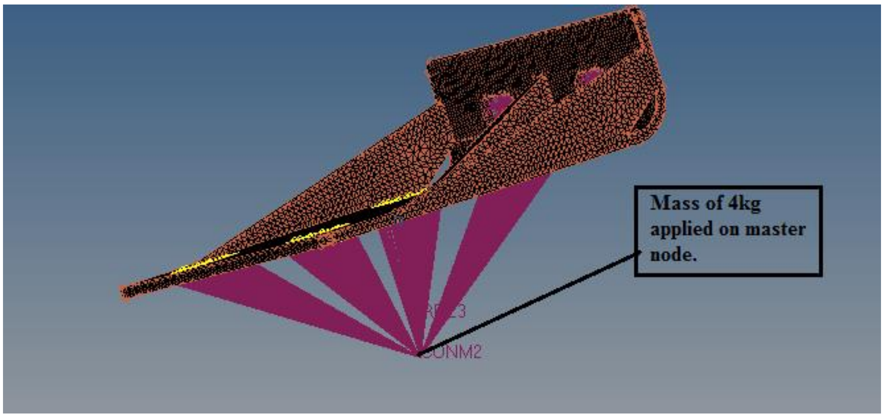


Figure 4.13 RBE 3 element with CONM2 mass of 4 kg

The Load Collectors used to perform modal analysis are explained below:

(i) Load Collectors: two load collectors were used in this analysis *i.e,* for constraints and load.

a) EIGRL (load collector): EIGRL card image is used to perform real Eigen value analysis (vibration) It is used to evaluvate the natural frequencies and their mode shapes using Lanczos Method.

b) Constraints (load collector): In this load collector, nodes attached to vehicle are fixed along all directions.

(ii) After this load step was defined for normal modes.

(iii) Design variables were created.

(iv) Two responses were created one with response type volume fraction and other with type frequency.

(v) Constraints were created with volume fraction upper bond of 0.750 as shown in figure below which means that 75% present volume of design space will be used to get optimized layout of silencer mounting bracket.

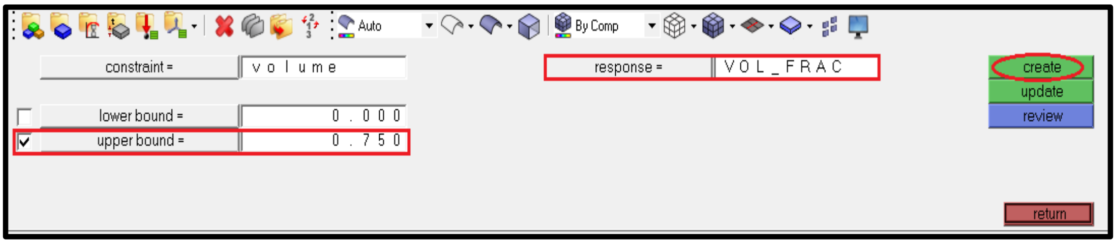


Figure 4.14 Creation of Constraints

(vi) After creation of constraints, objective function was defined with aim of maximizing frequency response to make bracket stiffer.

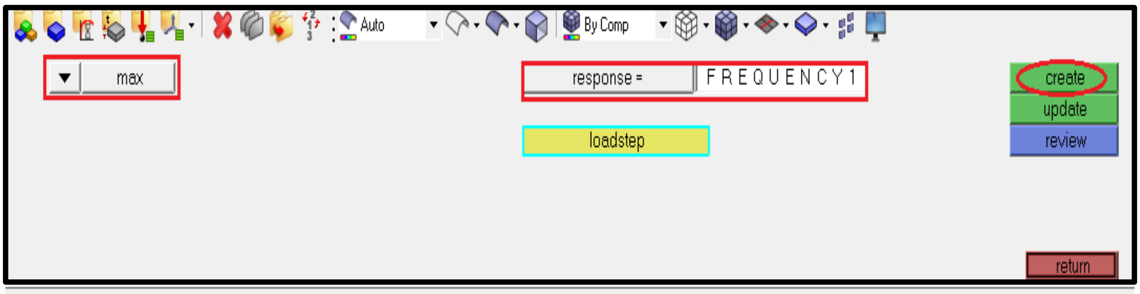


Figure 4.15 Defining objective

After performing the above steps, after several iterations were carried out Figure 4.16 shows the final result.

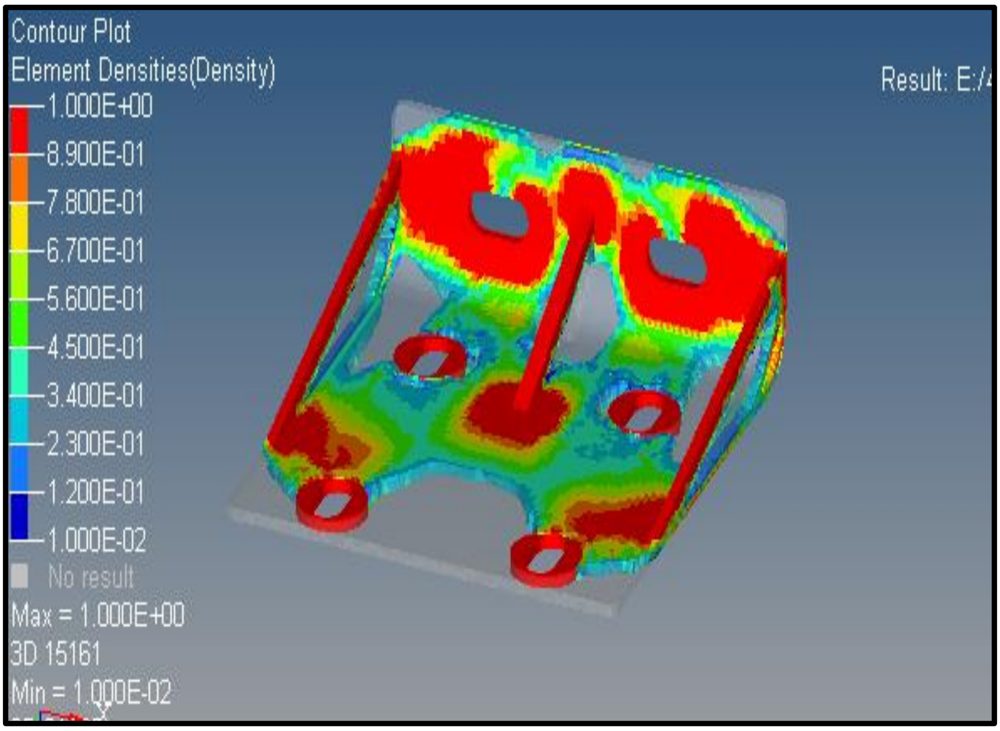


Figure 4.16 Final layout of Topology optimization on basis of modal analysis

Figure 4.17 shows the model browser for completing all necessary steps for topology optimization with help of modal analysis.

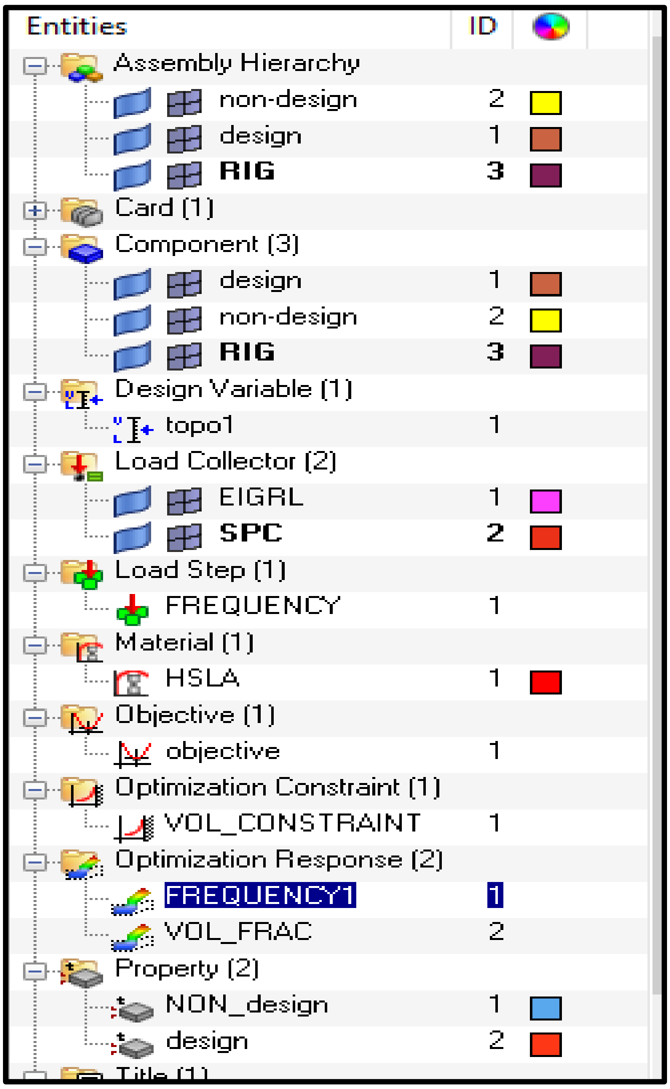


Figure 4.17 Model Browser of Topology optimization on basis of modal analysis

From figure 4.12 and 4.16 it can be seen that the optimized result is nearly same for optimization on linear static analysis basis and modal analysis basis. Finally both these files are converted in to IGES format with the post processing. A new model was made in pro-e with same dimensions for its further analysis Figure 4.17 shows the design of bracket after topology optimization.

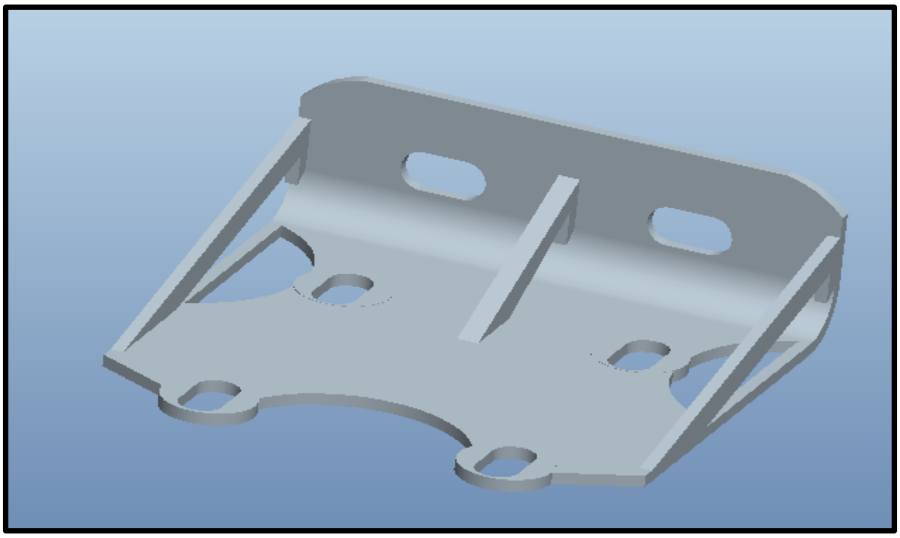


Figure 4.18 Design of bracket after topology optimization

The mass of the new model was measured to be 422.20 gram the original bracket mass measured earlier was 556.25 gram. It means percentage reduction of 24.1%, was obtained.

4.7 Static Analysis of Bracket after Topology Optimization

Static analysis was done on the bracket after topology optimization in ANSYS software. After applying boundary conditions and forces on meshed model of bracket the analysis was carried out in ANSYS software Figure 4.19 shows the boundary conditions and forces for static analysis of bracket.

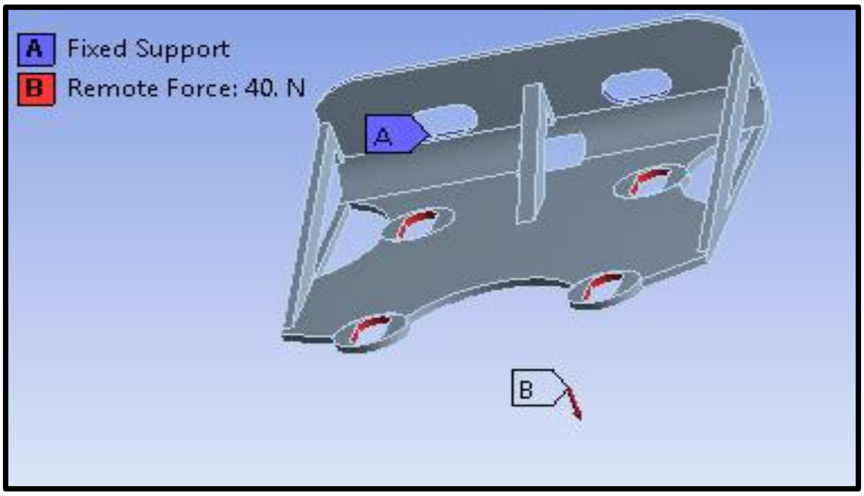


Figure 4.19 Fixed support and Remote force applied for static structural analysis after topology optimization

The result of total deformation and von-mises stress obtained after static structural analysis of optimised silencer mounting bracket with above forces and boundary conditions are shown Figure 4.20.

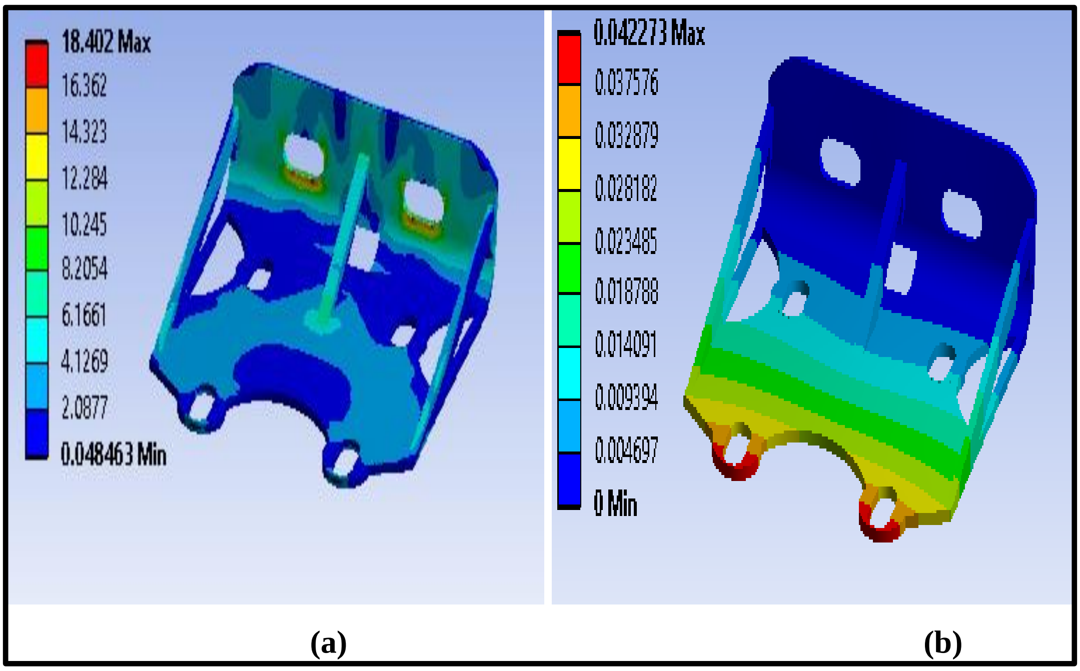


Figure 4.20 Results obtained after performing the static structural analysis: (a) Von mises stress (in MPa), (b) Total Deformation (in mm);

From above result we can see that modified bracket is in safe limits and the stress is less than the yield strength of material.

4.8 Dynamic Analysis of Bracket after Topology Optimization

Dynamic analysis was performed on bracket design obtained after topology optimization. Figure 4.21 shows the deformation and mode shapes corresponding to different natural frequencies.

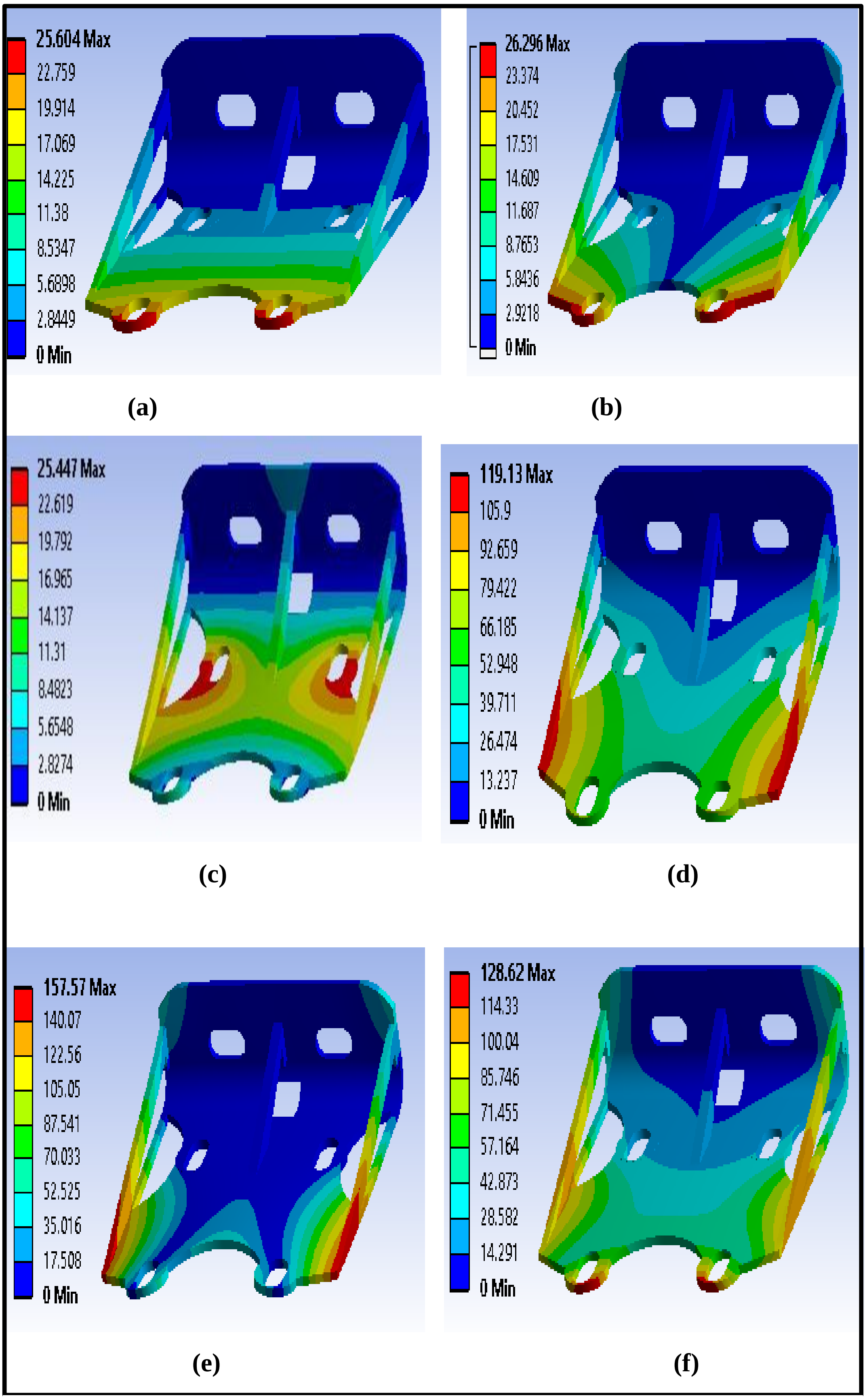
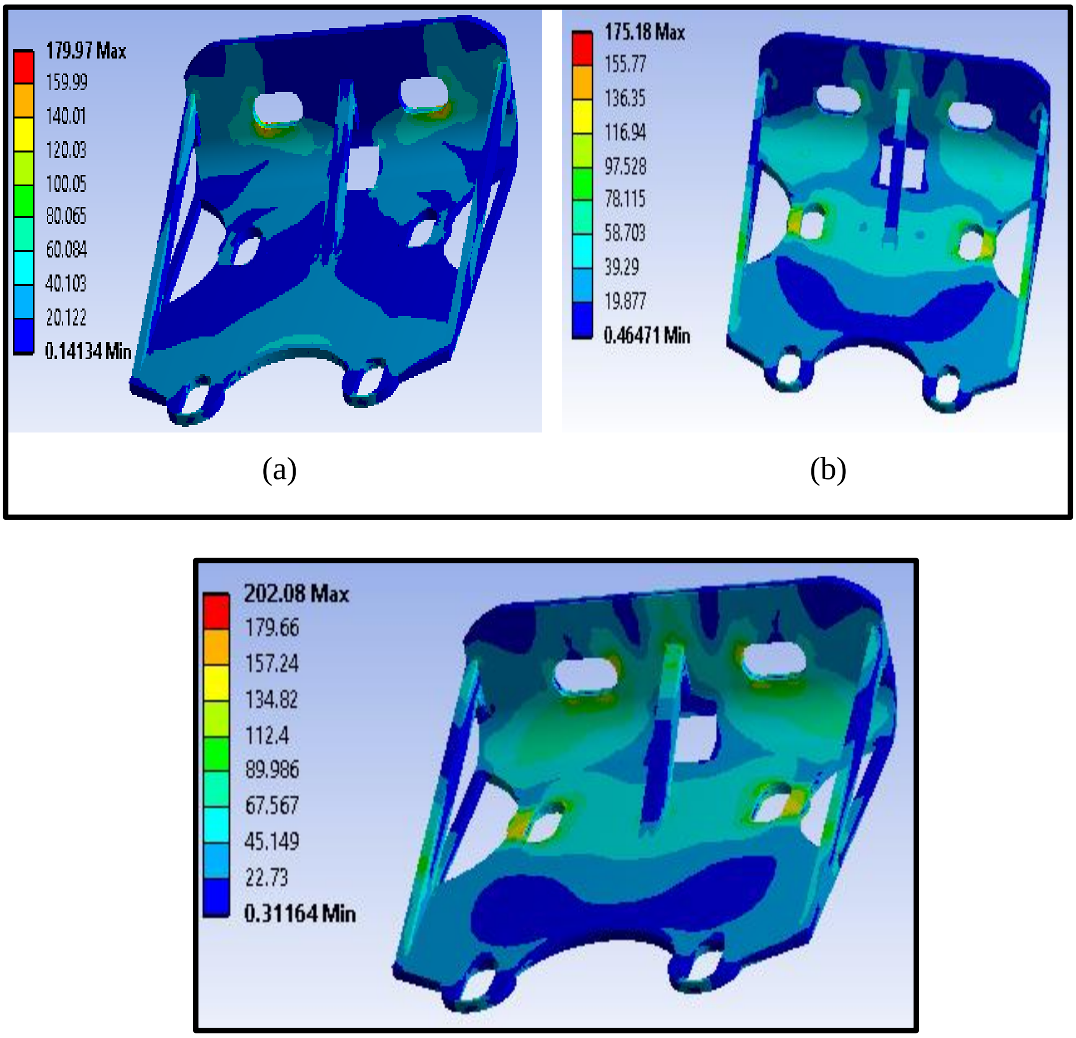


Figure 4.21 Mode Shapes for the bracket: (a) Mode shape 1 at 91.753 Hz; (b) Mode shape 2 at 173.53 Hz; (c) Mode shape 3 at 447.56 Hz; (d) Mode shape 4 at 947.67 Hz; (e) Mode shape 5 at 1505.8; (f) Mode shape 6 at 2169.6;

We can see from Figure 4.21 the value of natural frequency is more and improved as compared to original design of silencer mounting bracket which indicates that stiffness of bracket has increased.

After getting different mode shapes of bracket the frequency response analysis was performed in three directions (x, y, z). The figure 4.22 shows the stresses at first natural frequency 91.753Hz.



(a) (b)

(c)

Figure 4.22 Von- mises stresses at natural frequency of 91.753 Hz:

(a) Acceleration in x-axis (b) Acceleration in y-axis (c) Acceleration in z-axis

It can be clearly seen from Figure 4.22 that the stress is under permissible limit after topology optimization of bracket.

4.9 Topography Optimization

It is a type of shape optimization in which design region is defined and patteren of shape variable within that region is generated using optistruct. It is similar to Topology optimization except that shape variables are used rather than density variable.

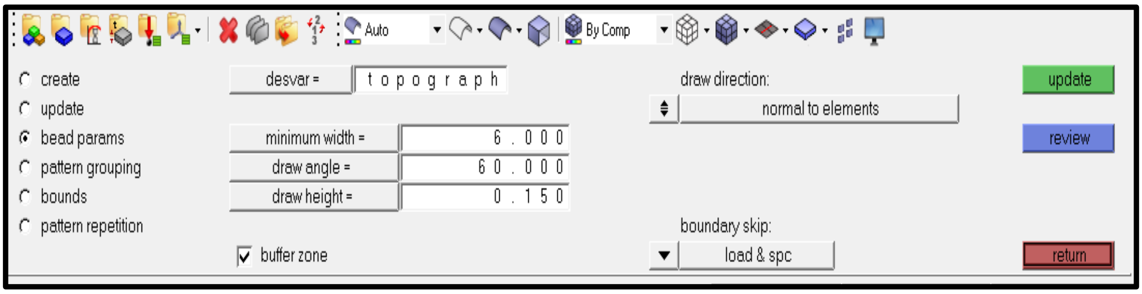


Figure 4.23: Topography with different parameters

After specifying above parameters of pattern grouping, bounds, pattern repetition. Response, objective and constraints were defined as in the case of topology optimization. Figure 4.24 of model browser tab shows the whole procedure for topography optimization.

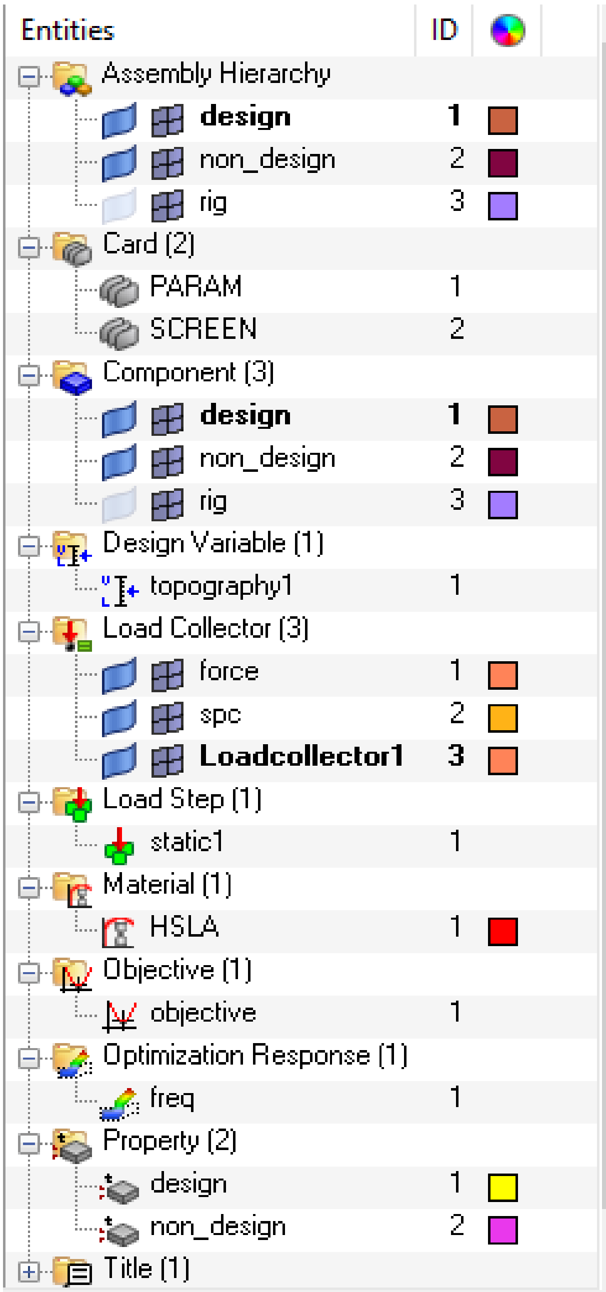


Figure 4.24 Model Browser of Topography Optimization

The final result was obtained after the last iteration. In final iteration the thickness of bracket is varied from different places. Figure 4.25 shows the final result of topography optimization.

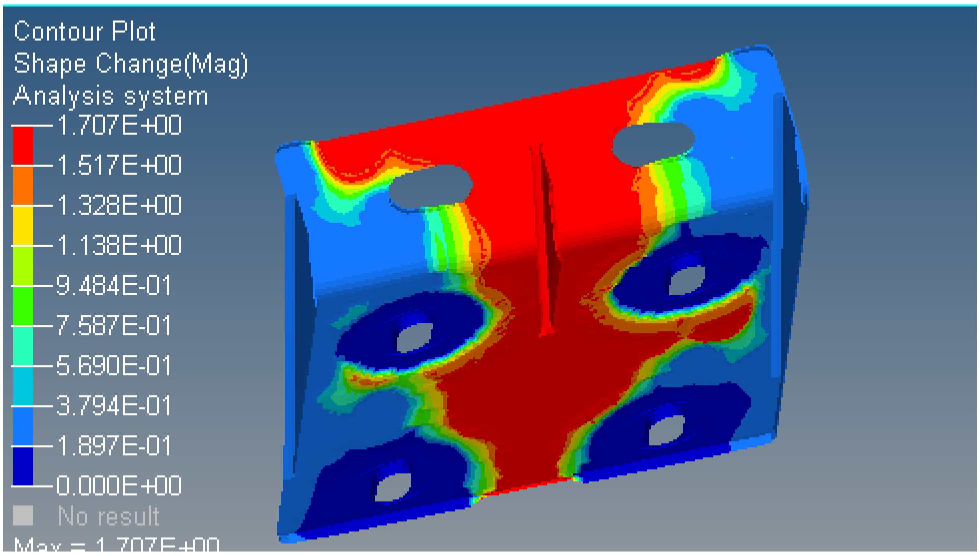


Figure 4.25 Final model after Topography optimization

The Figure 4.25 clearly shows that the central part of bracket which is shaded red has less thickness than area of bracket shown in blue. Finally these files are converted in to IGES format with the post processing. A new model was made in Pro-e with same dimensions for its further analysis Figure 4.26 shows the design of bracket after topography optimization. The mass of bracket was reduced to 473.10 gram from 556.25 gram. It means percentage reduction of 14.94%, was obtained.

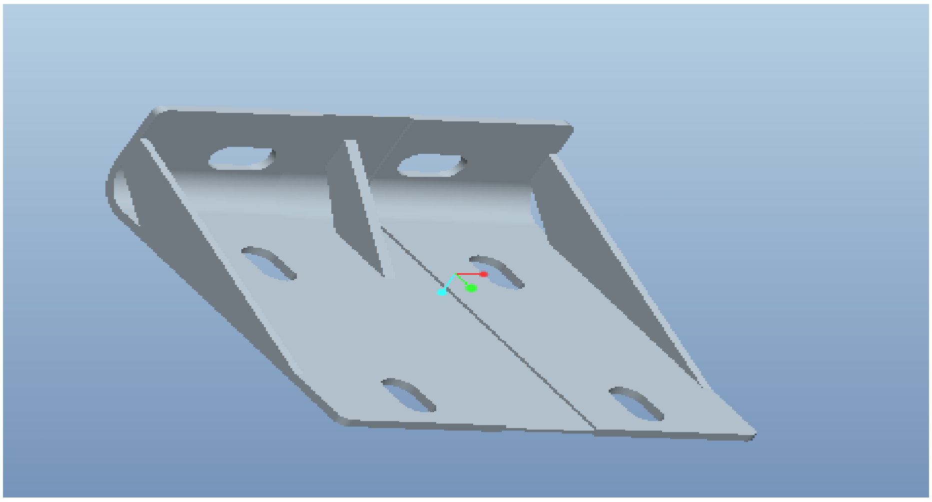


Figure 4.26 Design of bracket after topography optimization

4.9.1 Static Structural Analysis of Bracket after Topography Optimization

Static structural analysis was performed on the bracket after topography optimization. Analysis of optimised bracket was carried out in ANSYS software. Figure 4.27 shows the boundary conditions and forces for static analysis of optimised bracket.

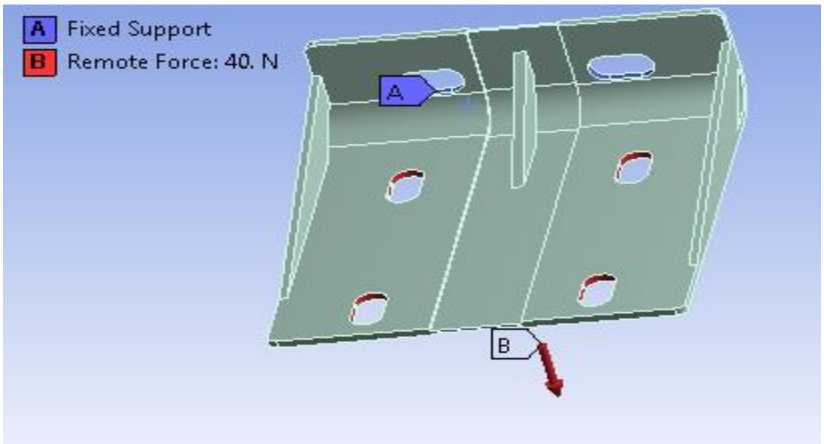
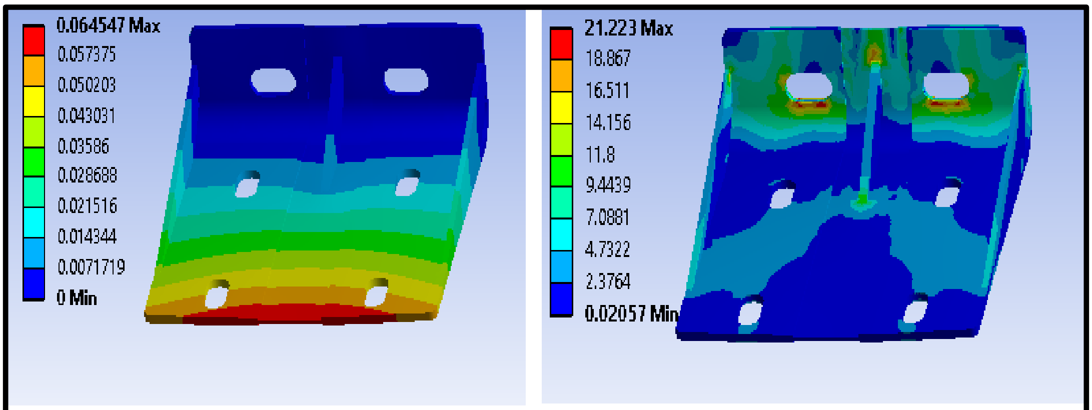


Figure 4.27 Fixed support and Remote force applied for static structural analysis after topography optimization

The result of total deformation and von-mises stress obtained after static structural analysis of silencer mounting bracket after topography optimization with above force and boundary conditions are shown below in Figure 4.28.



(a) **(b)**

Figure 4.28 Results obtained after performing the static structural analysis: (a) Total Deformation (in mm); (b) Von mises stress (in MPa)

Dynamic analysis was performed on bracket design obtained after topography optimization. Figure 4.29 shows the deformation and mode shapes corresponding to different natural frequencies.

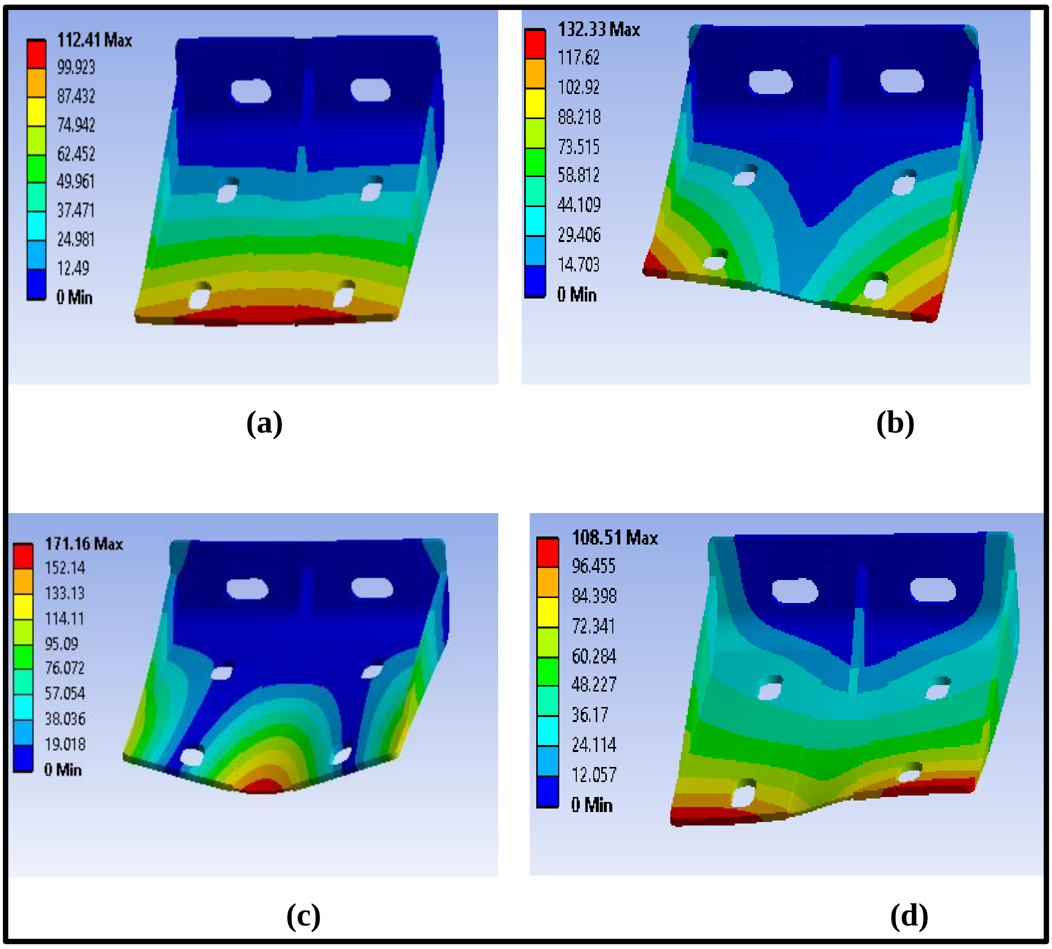


Figure 4.29 Mode Shapes for the bracket: (a) Mode shape 1 at 92.24 Hz; (b) Mode shape 2 at 161.85 Hz; (c) Mode shape 3 at 478.26 Hz; (d) Mode shape 4 at 900.33Hz;

It can be seen from the figure 4.29 the value of natural frequency is more and improved as compared to original design of silencer mounting bracket which indicates that stiffness of bracket has increased.

After modal analysis the frequency response analysis was performed in three direction x, y, z. Figure 4.30 shows the stresses at first natural frequency 92.24 Hz.

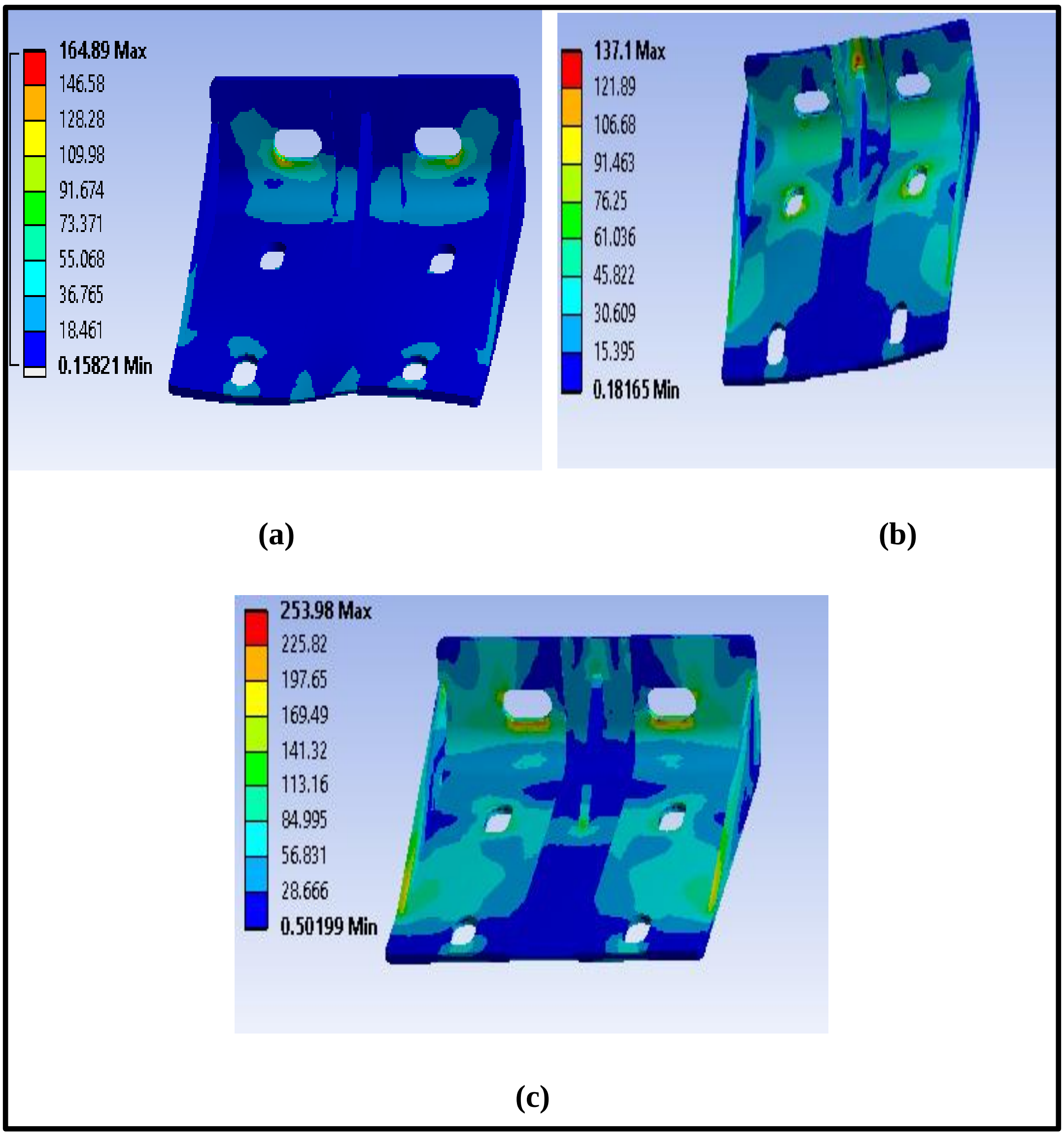


Figure 4.30 Von- Mises stresses at natural frequency

(a) Acceleration in x-axis (b) Acceleration in y-axis (c) Acceleration in z-axis

After dynamic analysis it can be seen that the stress produced in the topography optimised bracket is under the prescribed limit.

4.10 Shape optimization of bracket

Shape optimization is a method of optimization which increases or decreases the value of design variable like hole diameter, fillet radius thickness, etc. to achieve the defined objectives. In this case shape optimization was carried out with the help of ANSYS 15 software.

First of all static structural analysis was done on the bracket and output of static structural analysis was used as input for dynamic analysis. After dynamic analysis different parameters were defined which includes thickness of right, left and central rib and thickness of whole plate after these steps optimization was carried out with objective function to minimize weight. The constraint was defined that minimum fundamental frequency should be above 96 Hz so that the stiffness would not decrease.

4.10.1 Static and Dynamic Analysis after Shape Optimization

The static and dynamic analysis was performed after shape optimization. Figure 4.31 shows the stresses and deformation after static structural analysis of shape optimised silencer mounting bracket and Figure 4.32 shows deformation of bracket at different natural frequencies.

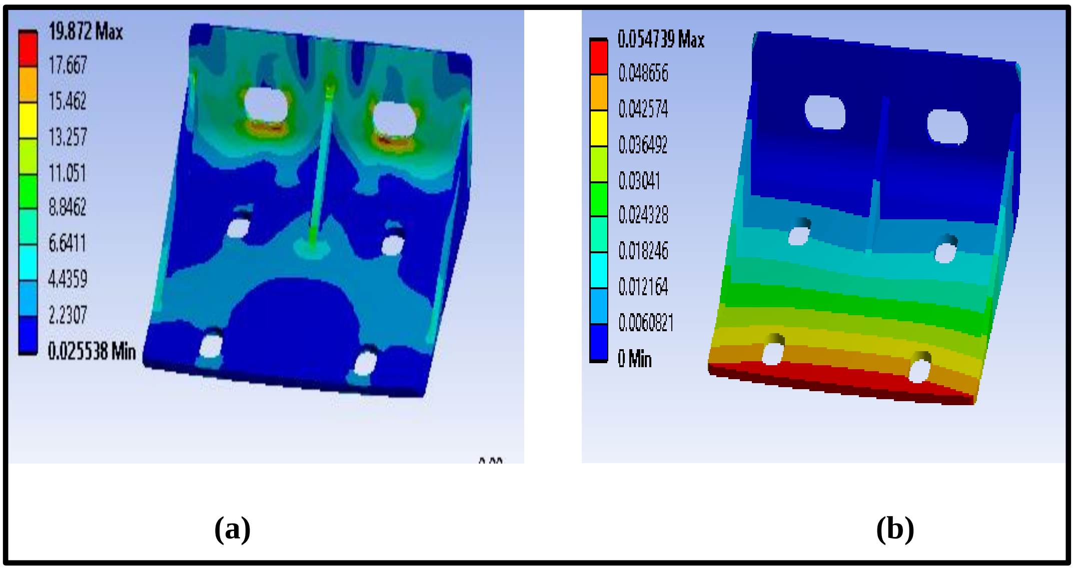


Figure 4.31 Results after Static Structural Analysis:

(a) Von-mises stress(in MPa); (b) Total deformation(in mm);

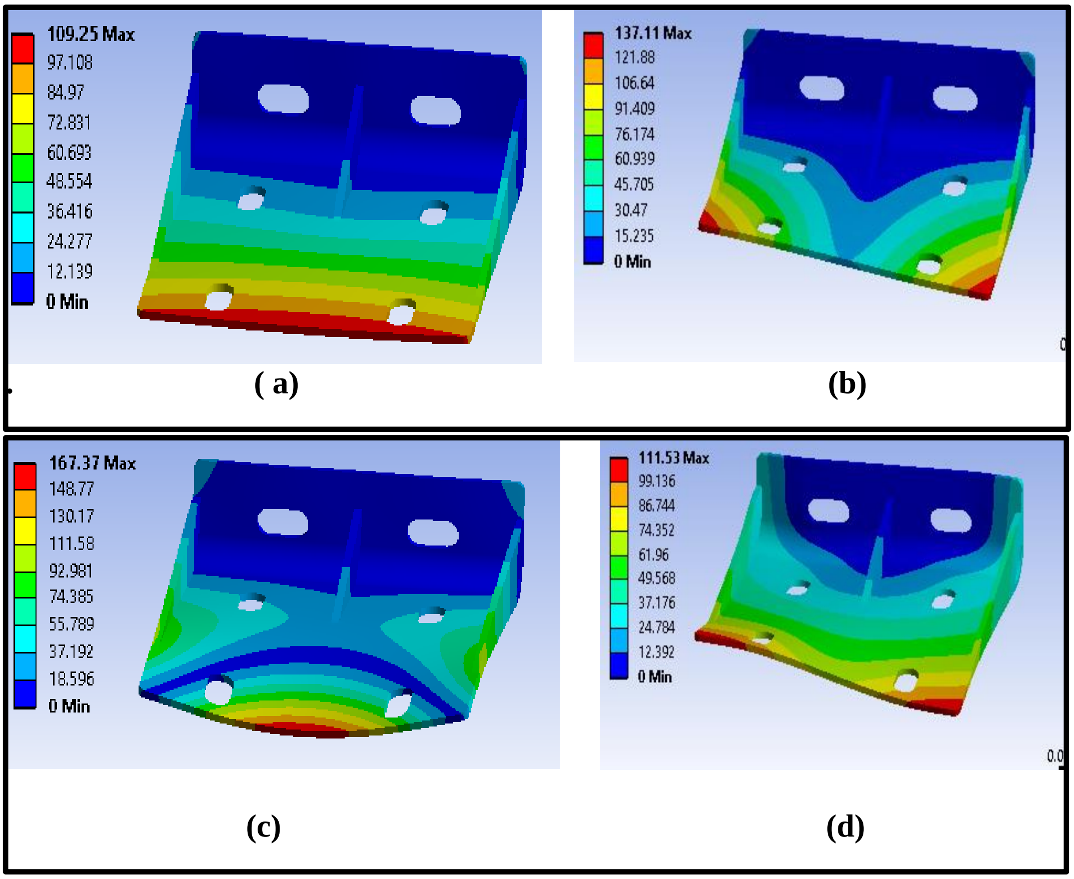


Figure 4.32 Total deformation (in mm) at different natural frequencies

(a) Mode shape 1 at 96.69 Hz ; (b) Mode shape 2 at 169.31Hz ;

(c) Mode shape 3 at 478.09 Hz; (d) Mode shape 4 at 837.25 Hz ;

From figure 4.32 we can see that the natural frequency is more than that of original design of bracket which means optimized design is much stiffer and isolated from vibration.

After this frequency response analysis was performed in x, y and z direction. Figure 4.33 shows the stresses at fundamental frequency.

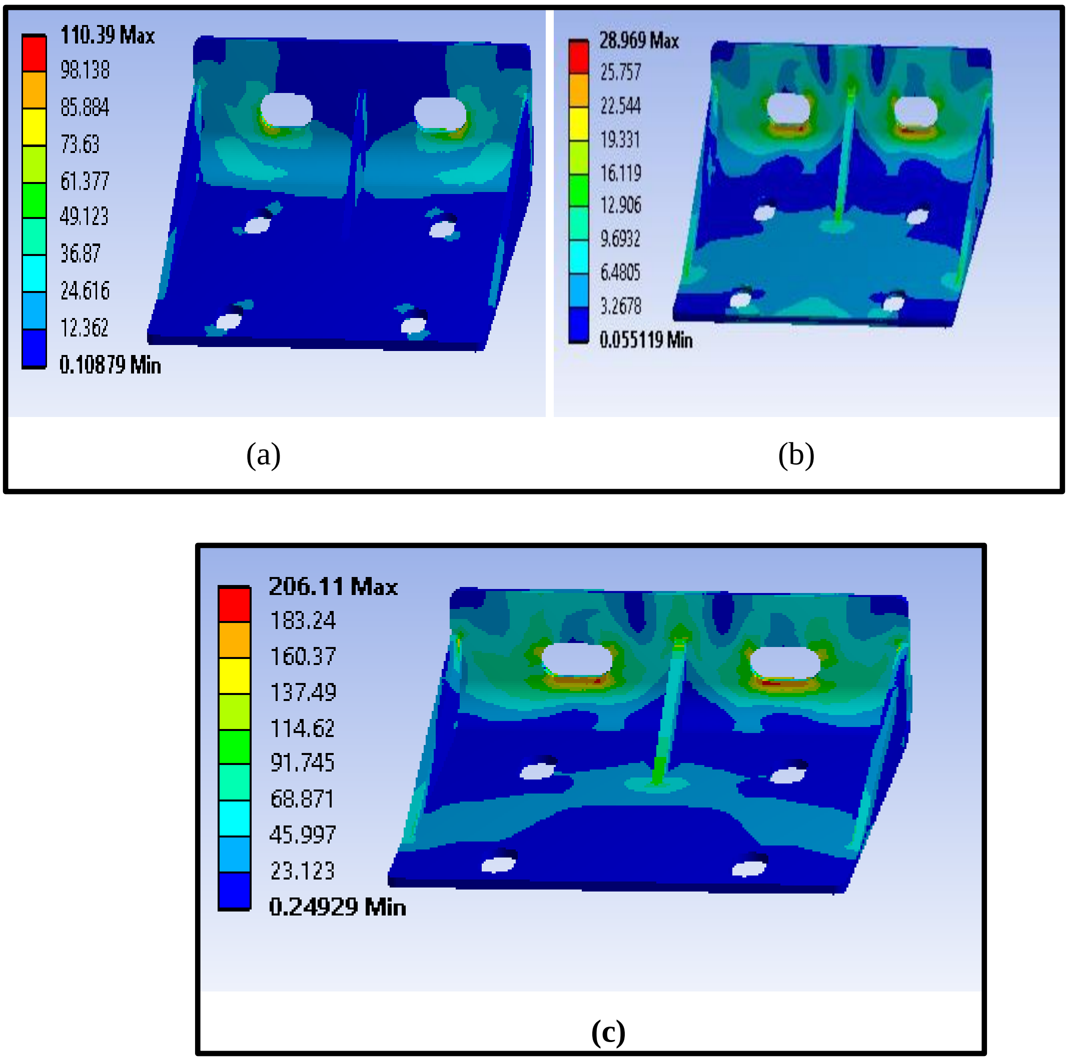


Figure 4.33 Von- mises stresses at fundamental frequency

(a) Acceleration in x-axis (b) Acceleration in y-axis (c) Acceleration in z-axis

It can be clearly seen from Figure 4.33 that von-mises stress is under prescribed limit so design is safe.

Table 4.1 Mass and Fundamental frequency comparison of original and optimized Bracket

Parameter	Original	After Topology Optimization of original bracket	After Topography Optimization of original bracket	After Shape optimization of original bracket
Mass	556.25g	422.20g	473.10g	493.09g
First natural frequency (From normal modes analysis)	78.832	91.753	92.237	96.69

From table 4.1 it is clearly seen that after each optimization weight was reduced and fundamental frequency was increased when compared with the original design of bracket. This means that stiffness of bracket was increased with reduction in mass.

CHAPTER 5 CONCLUSION

5.1 Conclusion

The use of Computer Aided Engineering as a tool which helps in visualising and comparing different designs has been illustrated. It was used to give optimised results in terms of mass and natural frequency for an automobile component.

Static structural and dynamic analysis was performed on a silencer mounting bracket of automobile using ANSYS software The results showed that there was a large scope to reduce the mass of bracket as stresses induced in the bracket were very less. The original bracket was optimised using topography, topology and shape optimization. Topology and topography optimization was done in Hypermesh (Optistruct), structural design and shape optimization was done using ANSYS. The reduction in mass achieved was 24.09% in case of topology optimization, 14.94% in case of Topography optimization and 11.35% in case of shape optimization.

So the presented CAE tools can be used for optimizing products used in different applications by design engineers, product managers and other stake holders, thereby reducing cost, effort and time in product life cycle.

.

5.2 Scope for Future Work

a) Other mounting parts of automobile can be optimised using the same procedure, presented in this work

b) Practical validation of this method can be done by vibration testing of bracket using electrodynamic shaker.

c) Fatigue and endurance testing can be performed on the bracket.

d) Transient response analysis for the bracket can also be explored.

REFERENCES

[1] Anton Olason and Daniel Tidman (2011)," Methodology for Topology and Shape Optimization in the Design Process", Master’s Thesis in Solid and Fluid Mechanic, Chalmers University of Technology, Sweden.

[2] Juan Pablo Leiva (2011), "Structural Optimization Methods and Techniques to Design Efficient Car Bodies", Vanderplaats Research and Development, Inc.

[3] Youngwoo Choi, Dohyun Jung, Kyoungchun Ham and Sungin Bae (2011), “A Study on the Accelerated Vibration Endurance Tests for Battery Fixing Bracket in Electrically Driven Vehicles”, Procedia Engineering, vol. 10, pp. 851–856.

[4] C. Wijaya, M. C. Yoon and B. T. Kim (2012), “Numerical Investigation on the Dynamic Characteristics of an Automotive A/C Hose”, International Journal of Automotive Technology, vol. 13, pp. 433–440.

[5] Uwe Schramm, Harold Thomas and Ming Zhou (2002), “Manufacturing Considerations and Structural Optimization for Automotive Components”, SAE Technical Paper, Doi: 10.4271/2002-01-1242

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

[7] Hong Suk Chang (2006), “A Study on the Analysis Method for Optimizing Mounting Brackets”, SAE Technical Paper, Doi:10.4271/2006-01-1480.

[8] K.S. Raghavan, T. Howard, J. Buttles and N. Law (2009), “Weight Reduction of Structural Vibration Isolation HydroMount Bracket through Design Analysis and Use of Advanced High Strength Steels”, SAE Technical Paper, doi: 10.4271/2009-01-1230

[9] Denghong Xiao,Xiandong Liu,Wenhua Du,Junyuan Wang and Tian He (2012), “Application of Topology Optimization to Design an Electric Bicycle Main Frame”, Struct Multidisc Optim, vol. 46,pp. 913–929.

[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] S. Laxman and R.Mohan (2007), “Structural Optimization: Achieving a Robust and Light-Weight Design of Automotive Components”, SAE Technical Paper, Doi: 10.4271/2007-01-0794.

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

[13] Bo Tan, Yu Yang, Jun Huang, Wenhui Liu and Dongqing Zhang (2015), “Structural Optimization for Heavy Truck Proppler Bracket”, SAE Technical Paper, Doi: 10.4271/2015-01-0638

[14] Takanobu Sailto,Jiro Hiramoto and Toshiaki Urabe (2014), “Development of Optimization Method for Automotive Parts and Structures”, SAE Technical Paper, Doi: 10.4271/2014-01-0410

[15] Vibhay Kumar and Gopala Uppala (2013), “Correlation Study of Oil Strainer Mounting Bracket for Dynamic Loading”, SAE Technical Paper, Doi: 10.4271/2013-01-2782

[16] ASM International the material information society, Publication title-Alloying understanding the basics Product code # 06117G: 2001; Pages: 647; ISBN: 978-0-87170-744-4.

[17] http://www.altairhyperworks.in/product/HyperMesh. (Accessed on 04/05/2016)

[18] Yasuo O., Nagashima H. and Tanahashi (1995), “Vehicle technology handbook tests and evaluation”, Society of Automotive engineers of Japan.

[19] Jin Zhu, Gary W. Krutz and Kamyar Haghighi (1988), “Dynamic Analysis of an Engine Chassis Mounting Bracket using the Finite Element Method”, SAE Technical Paper, Doi: 10.4271/881854

[20] Kendra Eads, Kamyar Haghighi ,Han- Jun Kim and John M.Grace (2000), “ Finite Element Optimization of an Exhaust System”, SAE Technical Paper, Doi: 10.4271/2000-01-0117.

[21] Junichi Fukushima, Katsuyuki Suzuki and Noboru Kikuchi (1992), “Shape and Topology Optmization of a Car Body with Mutiple Load Conditions”, SAE Technical Paper, Doi: 10.4271/920777.

[22] Basem Alzahabi, Scott C. Simon and Logesh Kumar Natarajan (2003), “Optimization of Transmission Mounting Bracket”, SAE Technical Paper, Doi:10.4271/2003-01-1460.

[23] Nitin S Gokhale (2008), Sanjay S Deshpande, Sanjeev V Bedekar and Anand N Thite, “Practical Finite Element Analysis”, Finite to Infinite, India, 2008.

[24] http://altairenlighten.com/2011/12/sizing. (Accessed on 04/06/2016)

[25] Osher S., Sethian J. (1988). Front propogation with curvature-dependent speed:Algorithms based on Hamilton-Jacobi formulations, J Comp. Physics Vol 79 12-49

[26] Wang M. (2004). Physical Modeling and Optimization of Heterogenous Solid. Level Set Methods, 2004 International Conference on Manufacturing Automation.

[27] Allaire G., Jouve F., Toader A. (2004). Structural optimization using sensitivity analysis and a level set method, J. Comp. Phys., Vol 194/1, 363-393

[28] Senthilnathan Subbiah, O.P. Singh, Srikanth K. Mohan and Arockia P. Jeyaraj (2011), “Effect of Muffler Mounting Bracket Designs on Durability”, *Engineering Failure Analysis*, vol. 18,pp. 1094–1107

[29] M D Rao, K J Wirkner and S Gruenberg (2004), “Dynamic Charactersistics of Automotive Exhaust Isolators”, Proceedings of the Institution of

Mechanical Engineers, Part D: Journal of Automobile Engineering, vol. 218, pp. 891–900.