OPTIMIZATION OF COMPRESSOR MOUNTING BRACKET OF AN INDIAN PASSENGER CAR

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

MASTER OF ENGINEERING

IN

CAD/CAM ENGINEERING

Submitted By

SACHIN KALSI

Roll No. 801381018

Under the Guidance of

Dr. JASWINDER SINGH SAINI	**Mr. DALJEET SINGH**
Assistant Professor,	**Assistant Professor,**
Mech. Engg. Deptt.,	**Mech. Engg. Deptt.,**
TU, Patiala.	**TU, Patiala.**

MECHANICAL ENGINEERING DEPARTMENT

THAPAR UNIVERSITY

PATIALA-147004, INDIA

July-2015

DECLARATION

I hereby declare that the work in this dissertation report entitled "**OPTIMIZATION OF COMPRESSOR MOUNTING BRACKET FOR AN INDIAN PASSENGER CAR**" 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 **Dr. JASWINDER SINGH SAINI,** Assistant Professor, Mechanical Engineering Department, Thapar University, Patiala and **Mr. DALJEET SINGH,** Assistant Professor, Mechanical Engineering Department, Thapar University, Patiala, during **July 2014 to June 2015.** 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.

14/7/2015

(Sachin Kalsi)
Reg. No. 801381018

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

14/7/15

Dr. JASWINDER SINGH SAINI
Assistant Professor,
Mech. Engg. Deptt,
Thapar University,
Patiala-147004.

14/7/15

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 guides ***Dr. Jaswinder Singh Saini*** *and* ***Mr. Daljeet Singh*** *for their 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. Their 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.

Sachin Kalsi

Reg. No.: 801381018

ABSTRACT

With the increase in competition among the industries, there is a need for timely and cost effective solution for the problems based on the design of any component. The different CAE tools come to the rescue of the designer.

In the present work, the CAE tools are used for the optimization of the compressor mounting bracket used in automobile. Both static and dynamic analysis is done for 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. The optimized design given by CAE tool is then validated using experimental setup. The CAE and experimental results shows very good correlation between the two.

INDEX

CONTENTS **Page No.**

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1 INTRODUCTION

1.1 Computer Aided Engineering

The design and development of an automobile is a big and complex task for a company or a designer team to fulfil all the requirements of a vehicle according to customer need and different standards adopted by various authorities in different countries. It is an extremely market-oriented task with the instantaneous and regular feedback from the public. The continual development, improvement in quality and incorporation more additional features without much variation in the market price have proved to be very effective in the highly competitive field of automobiles. To reduce the cost, the designer always tries to minimize labour, raw material and other costs that are required to manufacture an automobile. Computer Aided Engineering (CAE) comes to the rescue of the designer for cost effective and timely solution. CAE tools have made design and analysis faster and cheaper, reducing time to market, and making it possible to produce parts at lesser cost in the highly competitive automobile industry.

The benefits of CAE include:

(i) Reduction of product development cost and time, with improved quality of product and its durability.

(ii) Decisions of design can be easily made that are based on their impact on performance.

(iii) Designs of product can be evaluated and much refined using computer simulations rather than physical prototype testing of that product which as a result saves money and time.

(iv) It provides performance insights earlier in the development process stage, the time when design changes are less expensive to make.

(v) It helps engineering teams to manage any risk and understand the performance implications of their design of a product.

(vi) When properly integrated into product and manufacturing development, CAE can enable earlier problem resolution, which reduces the costs associated with the product lifecycle.

(vii) Graphical presentation of results.
(viii) Analyses are simplified through Graphical User Interface (GUI).

The different types of analyses that can be done using CAE tools are presented in the following sections.

1.2 Static Structural Analysis

Static structural analysis determines the effects of steady load conditions on a component. It ignores inertia and damping effects that are caused due to time-varying loads. But, it can include steady inertia loads *i.e.,* gravity and rotational velocity. In this analysis, the time varying loads can be approximated as static equivalent loads.

1.2.1 Type of Loads Applied

The types of loads that can be applied during static structural analysis are:

(i) Externally applied pressures and forces
(ii) Rotational velocity or gravity that are steady state inertia forces
(iii) Non zero displacement
(iv) Temperature effect

1.2.2 Types of Static Structural Analysis

Depending upon the behaviour of a material, loading and environmental conditions, there are mainly two types of static structural analysis that can be performed. They are:

1.2.2.1 Linear static structural analysis

It represents the most basic analysis. The meaning of linear means that computationally stress and displacement vary linearly with each other *i.e.,* displacement is linearly proportional to the applied load. It follows Hooke's law. Linear static structural analysis follows equation (1.1).

$$[K]\{u\} = \{f\} \tag{1.1}$$

where, [K] is the stiffness matrix of a system, {u} is the displacement vector and {f} is the applied forces vector. Using equation (1.1), stresses, strains and other secondary variables are calculated.

1.2.2.2 Non-linear static structural analysis

When the relationship between applied loads to response is not linearly proportional, then that type of analysis is known as non-linear static structural analysis. This type of analysis is mostly used in problems in which the material behaviour or geometry of the structures is not linear. The non-linear behaviour occurs due to some reasons that are given below:

a) Non-linearity due to boundary

It arises when the boundary conditions in a finite element model changes during analysis. The boundary conditions could be added or removed from the model due to boundary non-linearity as the analysis progresses. This type of non linearity involves set of contacts in the model which get engaged or disengaged as a response to applied loads **[1].**

b) Non-linearity due to geometry

It arises when the stiffness of the structure changes due to geometric deformations. The different non-linearities due to geometry are given below:

(i) Large strain
(ii) Large deflection or large rotation
(iii) Stress stiffening
(iv) Spin softening

c) Non-linearity due to material

Non-linear stress strain relationships are a common cause of non-linear nature of structure. This is due to properties of material, environmental parameters like temperature, load history like elastoplastic response and time for which load is applied.

For non-linear problems, mostly Newton Raphson approach is applied. In this, the load is divided into load increments using the load steps.

1.3 Dynamic Analysis

When a load on structure varies with time, then it is assumed that the response of a structure will vary with time. In such type of cases, dynamic analysis plays a vital role in finding the response of a structure. If the frequency of load is low as compared to natural frequency of a structure then static analysis of a structure is assumed to be carried out. But, this assumption is valid when the frequency is one third of the natural frequency of the structure. Depending upon the type of loading and required response of the structure different types of dynamic analysis are carried out to get the better results.

Equation (1.2) gives the dynamic response of a structure or component.

$$[M]\{\ddot{D}\} + [C]\{\dot{D}\} + [K]\{D\} = \{F\} \tag{1.2}$$

where, [M] is the structural mass, $\{\ddot{D}\}$ is the nodal acceleration vector, [C] is the structural or component damping matrix, $\{\dot{D}\}$ is the node velocity vector, [K] is the structure stiffness matrix, {D} is the node displacement vector and {F} is the applied time varying nodal load vector. Equation (1.2) consists of set of differential equations in matrix form to get a dynamic response of a structure or component modelled with a finite number of degrees of freedom.

Equation (1.2) gives the response varying with time at each point of a structure by considering inertia forces as well as forces due to damping. The damping forces are product of damping coefficient and velocity, whereas inertia forces are product of mass and acceleration. With the increment of time, the solution for equation (1.2) will generate thousands of static solutions to get a complete time response for the structure.

1.3.1 Types of Dynamic Analysis

Following are the different type of dynamic analysis that are normally carried out according to the conditions and loads applied.

1.3.1.1 Modal analysis

In many of the engineering problems, vibration analysis is one of the main analyses to be carried out. It is known as eigenvalue analysis. It is used to obtain natural frequencies and deformations at the natural frequencies. The eigenvalues give natural frequencies and eigenvectors give the mode shapes that are deformation in the structure due to natural frequencies. These are un-damped free vibrations that are generated due to initial disturbance of the structure from the static equilibrium state or position of it. By removing the inertia forces and damping forces in the general dynamic equation (1.2), the solution is obtained. Thereafter, it is assumed that the structure is subjected to sinusoidal functions of the peak amplitude. Let the displacement, {D} is of the form as shown in equation (1.3).

$$\{D\} = \{A\} \sin \omega t \tag{1.3}$$

where, A is the amplitude of the displacement and ω is a circular frequency. Here, $\{A\}$ is an eigenvector and is known as mode shape.

By differentiating and solving equation (1.3); equation (1.4) is obtained.

$$([K] - \lambda[M])\{A\} = \{0\} \tag{1.4}$$

where, λ corresponds to $\omega^2$and {A} associated with every value of λ. The natural frequency is obtained from the value of eigenvalues which depends upon the number of degree of freedom. Normally, first few eigenvalues are considered because the finite element model gives the approximation of the structure. Therefore, the higher eigenvalues and eigenvectors that is mode shapes are mostly neglected. The natural frequency of a structure is given by equation (1.5).

$$f_i = \frac{\omega_i}{2\pi} \tag{1.5}$$

where, f_i is i^{th} natural frequency, ω_i is $\sqrt{\lambda_i}$.

The linear combination of all of the normal modes and displacements gives the corresponding deformation or vector displacements of a structure under forced and free vibrations.

1.3.1.2 Modal frequency response analysis

It is used to compute the response of a structure when steady state oscillations are applied as excitation. It includes rotary machinery, helicopter blades, and supports of a component having vibrations. In this, excitation is defined in the frequency domain and the applied forces are known as forcing frequencies. The forces can be enforced motions or applied forces. The frequency of the response to the harmonic input is also harmonic and it occurs at the same frequency. The function of force is given by equation (1.6).

$$\{F\} = \{F_O\}e^{i\omega t} \tag{1.6}$$

where, F_O is the peak force amplitude, ω is the harmonic frequency.

The general equation of dynamic motion is given by equation (1.7).

$$(-\omega^2[M] + i\omega[C] + [K])\{D_O\} = \{F_O\} \tag{1.7}$$

1.3.1.3 Transient response analysis

If the input is not harmonic and changes with time *i.e.*, it is a function of time, then the transient response analysis is performed. In this, general equation is solved by considering the damping and inertia effects on the time scale loading. This type of analysis determines the displacements, strains and stresses that are varying with time. To perform this type of analysis, the following two methods are used.

a) Direct method

It involves solving the system of equations by direct integration approach. It requires number of time interval steps, with a complete solution at each step. It increases the computational time for complex problems.

b) Modal superposition method

It assumes that the response of the structure or component can be represented by lower natural frequencies. The complete response from the structure is the summation of the correct fractions of the mode shapes having low natural frequencies *i.e.*, transformation of the equation from nodal coordinates to a set of modal coordinates. This method easily uncouples the equations and gives the efficient solution in a short interval of time. But, the method gives the approximate solution to the problem.

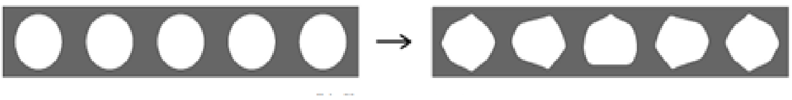
1.4 Optimization

Optimization is a technique used to get the best possible solution under some given or provided constraints and circumstances. Example of optimization can be optimal route followed by a salesman or in structural optimization, to find out the optimized distribution of material to satisfy the provided requirements. The optimization problem is generated and solved by engineers from their past experiences and subject knowledge.

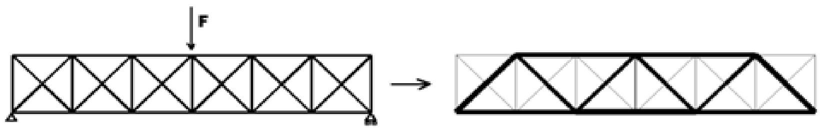
The objective function for the problem of optimization consists of the parameters that are to be maximized or minimized. The optimal solution obtained depends upon some design variables that can be expressed with numerical value.

Mathematically, the optimization problem can be formulated as minimizing (or maximizing by changing the sign of objective function to negative) the objective function that is subject to constraints.

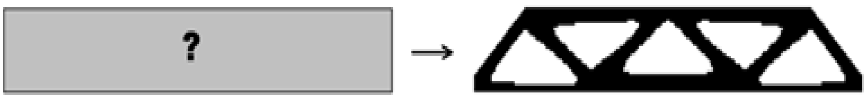
1.4.1 Types of Optimization Methods



(a) Shape Optimization



(b) Sizing Optimization



(c) Topology Optimization

Figure 1.1: Different type of optimization methods [2]

The different optimization methods available are given below:

a) Shape optimization

This optimization method considers the specified design parameters and varies them until the design responses and constraints are fulfilled. It changes the chamfers, fillets, radius, material thickness etc. Its algorithm doesn't include the removal of holes etc. Shape optimization is shown in Figure 1.1 (a).

b) Size optimization

It defines the ideal parameters of a component *i.e.*, dimensions of cross-section, material properties and thickness. 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 **[45]**. Sizing optimization is shown in Figure 1.1 (b).

c) Topology optimization

It is a main and powerful technique of optimization designed to evaluate and expand the space of solution and increase in creativity both at macro and micro scale. The software gives an optimal design with the given boundary conditions, design responses, loads, design space and constraints. With topology optimization, the shape and number of holes in a structure are not known. The disadvantage of topology optimization is that it provides a non-smooth product design which is further taken to estimate and modify it accordingly. Topology optimization is shown in Figure 1.1(c).

d) Topography optimization

This optimization is similar to topology optimization technique, but it is concerned with carrying 2D element that is offset from the component mid- plane. It is mostly used on sheet metal parts or components. In this, keeping the element thickness constant, but corresponding varying the surface topography, the optimal solution can be found out.

e) Topometry optimization

It is a general case of sizing optimization. In this technique each element is designed independently as compared to sizing optimization technique in which all elements that are associated are designed with the same values.

f) Freeform optimization

It is a special case of shape optimization. In this, the program splits any given perturbation into multiple perturbations based on grids. This results in increase of the design space variability as compared to traditional shape optimization. This type of optimization can be applied to any type of element. It is mainly used to design ribs on solid parts or structures. Also, used to design patterns of bead that can be constant or have variable height.

The implementation of different optimization techniques depends upon the type of manufacturing method used in a component. Table 1.1 shows the different optimization techniques preferred for different manufacturing methods.

Table 1.1: Different types of optimization methods [4]

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

CHAPTER 2 LITERATURE REVIEW

Number of researchers has worked on the analysis and optimization of different automobile components. The following section gives the review of such work.

2.1 Literature Review

Zhu *et al.* **[5]** analyzed the dynamic behaviour of bracket mounted on engine chassis using finite element method approach. The natural frequencies and mode shapes were evaluated under boundary conditions as per the requirements of an engine mount *i.e.*, the influence of C-section edges, bolt holes, ribs and boundary constraints was taken into account during analysis on the engine mount.

Fukushima *et al.* **[6]** used the concept of shape and topology optimization using homogenization method to design various car body problems under multiple loading conditions. The objective was to minimize the compliance with the constraint of certain material. The technique of optimization was also extended to three dimensional solids and shells. The problems of car body design with the requirements of strength and validity was solved using topology optimization.

Eads *et al.* **[7]** used finite element method to optimize the weight and cost of the exhaust design. Both static and dynamic loading conditions were used to solve the problem. The sinusoidal displacements were applied at the support points. The gravitational load was applied on the exhaust.

There was 22% reduction in weight under dynamic analysis and 25% reduction in weight of exhaust system in static condition.

Krishna [8] performed sizing, shape and topography optimization to reduce the weight of a jounce bumper bracket. The comparison of results obtained from each optimization was done. Each optimization method was applied independently of one another.

It was found that the topography optimized bracket was best in design with the advantage of additional manufacturing ease.

Schramm *et al.* **[9]** presented the techniques used in topology optimization to reduce the obstacles in manufacturing of a product. Manufacturing constraints were introduced which were helpful in manufacturability of a component.

It was concluded that the use of topology optimization reduced the number of spot welds in a vehicle as well as was used with the manufacturing constraint to reduce the obstacles in manufacturing of components.

Alzahabi *et al.* **[10]** used Computer Aided Engineering to provide the optimized design of bracket which was having more stiffness, strength and less mass. It was optimized by taking into consideration the constraints of volume available in transmission mounted in power train system. As the volume of the rubber was limited, therefore it was decided to maximize the stiffness of the bracket as much as possible.

It was found that the strength, stiffness and mass of the bracket were improved under the modal requirements.

Nelson [11] provided a practical application of die- direction constraints used in topology optimization on automotive lower control arm**.** The concept used in topology optimization in Altair Optistruct reduced the obstacles occurred in the manufacturability and also, improves the design of component obtained from topology optimization.

Lee *et al.* **[12]** performed only shape optimization of an air conditioner compressor mounting bracket used in passenger car. The objective of the optimization was to reduce the weight of a bracket with the design constraints as the first resonant frequency of the compressor assembly and the durability of the bracket. The compressor assembly that included compressor, connection elements and bracket were modelled using finite elements. The optimal design of a bracket had less mass with more strength.

Ko *et al.* **[13]** performed the concept of topology optimization on the steering column mounting bracket of the cowl cross member with the objective to decrease the weight and target vertical mode frequency to be increased. The cowl cross member was combined with the body in white (BIW) and then, the design sensitivity analysis was also performed to minimize the variables for optimization.

Porto *et al.* **[14]** performed topological and size optimization on the two components *i.e.*, rear cabin suspension bracket and frame air filter bracket. The objective function used in case of rear cabin suspension bracket was to minimize the compliance with the constraint of material volume. In case of air filter bracket, the material volume was chosen as the objective function with the constraint of natural frequency. In both the problems, static structural analysis was used as the loadstep for optimization.

Rao *et al*. [**15]** determined the dynamic characterization that included variation of stiffness and damping by changing the frequency generated through electro hydraulic and shaker excitation of the exhaust isolators used in automobile. Several samples of isolators with different materials were considered. Both swept sine and random excitations were used.

It was found that the results of stiffness and loss factor were very similar between shaker and electro hydraulic excitation except at low frequencies.

Chang [16] 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.

Leal *et al.* **[17]** performed an analysis to optimize the engine mount differential side of a commercial vehicle. The two different optimization techniques *i.e.,* try-error and topological method were implemented. The results obtained from topology optimization were considered to be more effective. Thereafter, the geometry from topological optimization undergoes more accurate optimization process in which the design variables were chosen to be geometrical shapes. The validity of the geometrical study was also carried out to adjust shape of geometry according to the manufacturing process.

Pan *et al.* **[18]** performed both shape and topology optimization for the design of an engine mount bracket to achieve more structural rigidity, more strength but with the reduction of mass. The die direction constraint and manufacturing constraints were also considered for topology optimization.

The mass of optimized design of bracket was reduced by 12%, whereas its strength and structural rigidity was improved by 50%.

Laxman and Mohan [19] performed topology optimization and gage optimization on three automotive components *i.e.,* transfer case assembly, step bar module and shock reinforcement assembly for a truck with the objective to reduce the mass without compromising of the structural requirements of the components *i.e.,* stiffness, modal frequencies, durability and crashworthiness.

Chang and Lee [20] performed topology optimization of compressor bracket. Base bracket for the topology optimization was modeled by considering the interference with the adjacent vehicle parts. The material of a bracket was chosen to be aluminum die casting material. Static and modal analysis was performed using CAE tools which were validated experimentally.

The mass of the optimized bracket increased marginally. The first natural frequency increased by 29 Hz. The averaged stress was reduced to 40% under the static load through succeeding structural analysis.

Loh *et al.* **[21]** performed fatigue analysis on air conditioner motor bracket under dynamic conditions as it undergoes huge vibration which may lead to its failure. Different alternative designs of bracket with different thickness were considered and evaluated.

The addition of ribs like support and increase in radius on the flange bracket improved the performance of the bracket.

Raghavan *et al.* **[22]** presented finite element analysis approach to reduce the weight of structural vibration isolation hydro-mount bracket using advanced high strength steel. The hydro-mount bracket was used to protect the fluid filled rubberized inner mount from shock loads. The static analysis was performed using ETA-VPG pre-processor. Solid elements were used for modelling. Alternate product design was proposed and corresponding weight savings were calculated. The durability and static performance was also verified experimentally.

The weight was reduced by 8% after making minor modifications in the design.

Jang *et al.* **[23]** performed topology and thickness optimization to minimize the mass of lightweight flatbed trailer design with the constraints to increase bending stiffness and torsional frequency.

The final design was lighter than the original design by 29% as well as its mean compliance decreased by 21% with uniform bending load. The torsional frequency was increased by 169%.

Cevik *et al.* **[24]** used shape optimization method to decrease the mass and moment of inertia of crankshaft but with least possible effect on the strength and stiffness under dynamic conditions. The multi body simulations were also used in the process of optimization.

The mass of crankshaft was reduced by 15%. The mass inertia was also reduced without any effect on the stress values.

Choi *et al.* **[25]** studied the accelerated vibration endurance tests of battery fixing bracket mounted in electrically driven vehicles. A single degree of freedom acceleration test method was introduced for battery fixing bracket. The accelerometers were used for measuring vibration history on real road and proving grounds.

The single axis acceleration test method was proposed to be more reliable and faster than six axis acceleration test method.

Cavazzuti *et al.* **[26]** used topology, topometry and size optimization approach to design the automotive chassis to reduce the weight of the chassis. The work was performed on rear central engine high performance vehicle chassis with Ferrari standards with the constraints of structural performance constraints. The numerical model of the vehicle chassis verifies the weight reduction when it was compared with the original chassis model.

Subbiah *et al.* **[27]** studied the effect of designs of muffler mounting bracket on durability. The cracks were observed at the weld location between brackets and engine cradle at an average distance of 10,000 km. Statistical analysis was done to identify the possible cause and effect of the failure of the bracket. The four main causes of the failure *i.e.,* man, method, material and design was identified using fishbone diagram. For structural finite element analysis, sheet metal parts were meshed using shell elements. The existing design was having 45^0 weld between the bush and bracket. The point mass was used for engine and muffler representation. ANSYS element mass 21 was modelled with rotary inertia. The 4g criteria were applied to muffler location. After structural analysis, the maximum stress was observed near the weld zone between engine cradle and the bracket. Different designs were proposed but the best design was achieved as per maximum stress generation and failure due to strain energy.

Wijaya *et al.* **[28]** performed the dynamic analysis *i.e.*, modal and transient analysis of an air conditioning system hose used in automobile using finite element method. The results were validated experimentally.

It was found that the swing and bending modes of the hose occurred in first six natural frequencies. In transient analysis, maximum stress occurred in the reinforced braid layers of hose components.

Bruggi and Duysinx [29] gave the formulation and procedure for the topology optimization of elastic structures with the aim to minimize the weight of a structure which was subjected to compliance and local stress constraints. The global constraints provided the expected stiffness to the optimized design. Comparison of pure stress based or pure compliance based strategies were also provided to point out the difference that arises in the optimal design with respect to the conventional approaches depending upon the assumed behaviour of the material.

Xiao *et al.* **[30]** implemented topology optimization in designing the main frame of electric bicycle under dynamic conditions to reduce the mass and to increase the structural stiffness. Flexible coupling dynamic model was also constructed. The optimization approach was found to be effective to develop the new design of the electric bicycle.

Park *et al.* **[31]** used the concept of shape optimization to maximize the isolation and fatigue life of rubber isolators used in automotive cooling modules. The CAE results were compared with experimental ones. The rubber isolators with optimal shapes for two different cooling modules were obtained.

Jain and Pavuluri [32] performed a non linear static analysis on bogie bracket assembly along with the contacts and bolt penetrations as it consists of bogie bracket, radius rods and leaf spring. The sensitivity analysis and surface contact behaviour between bogie suspension mounting bracket and fame were also studied and analyzed.

Good correlation in the results obtained experimentally and numerically was found.

Morales *et al.* **[33]** performed different optimization techniques on different vehicle structures using Altair Optistruct. The set of variables and constraints were chosen as per the problem.

Kumar and Uppala [34] studied the behaviour of oil strainer mounting bracket under dynamic conditions using Finite Element Analysis. Constrained Modal analysis as well as frequency response analysis was also performed for different exciting frequencies generated from the engine at different service loads. Modal superposition method was applied. The natural frequency along with stresses obtained from modal analysis was correlated with the experimental results.

Tsai and Cheng [35] proposed a technique to determine the material distribution of a structure to get the Eigen mode shapes as per the requirement for the problem that involved maximization of the fundamental Eigen frequency. The objective of the problem was achieved using the Solid Isotropic Method with Penalization (SIMP) for topology optimization. To maximize the fundamental natural frequency, weighted constraints were also added in the bound formulation. The Modal Assurance Criterion (MAC) as additional constraints was also added in the bound formulation optimization to get the desired Eigen modes in topology layout of the structure.

Salim *et al.* **[36]** analysed Toyota yaris sedan passenger seat model that was taken from National Crash Analysis Centre for the importance of federal motor vehicle safety standards 207/210 and provided additional optimum J-bracket which was cost effective, easy to manufacture and assemble. The said standard gave the requirements for seats, attachment assemblies and their installation to minimize the possibility of the failure on application of forces.

Sailto *et al.* **[37]** used the concept of topology optimization in automotive components that are made up of steel sheets with rectangular cross sections. The stiffness optimization was carried out for simple cylindrical model, floor model and full vehicle model. It was found that the topology optimization method was more effective in the optimization of said components.

Zakaria *et al.* **[38]** performed fatigue strain signal behaviour of different road surface conditions and its effect on the fatigue damage of an engine mount bracket. Two conditions were considered *i.e.*, residential area road surface and highway road surface. The strain signal behaviour was analyzed and further classified using both statistical and signal processing tools. The results were verified with the commercial finite element analysis software. Aluminium alloy AL6061-T6 was considered as a material of a bracket. The fatigue strain signals were collected using 1300 cc engine. The strain gauge was mounted on the most critical positions of the bracket and was determined using finite element analysis software. The fatigue strain signals of an engine mount bracket were done using different signal analysis approaches.

It was found that the fatigue strain signal behaviour was much affected by the road surface conditions.

Tan *et al.* **[39]** performed structural optimization on heavy truck propeller bracket with the consideration of dynamic response and dynamic stress on the bracket. The rigid flexible coupling and finite element analysis was considered for the propeller shaft, bracket, mount and frame assembly. The effects on the structure of the bracket with the variation in frequency were also studied.

The optimal design of propeller shaft bracket was presented which was validated experimentally.

2.2 Gaps in Literature

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

(i) Transient response analysis has not been performed on air conditioning and engine mounting brackets.

(ii) Non-linear static structural analysis has not been performed to see the behavior of material and structure under non- linear conditions.

(iii) Different structural optimization techniques like shape, size, tomography, topometry and freefoam have not been performed on number of components as seen in the literature.

(iv) Comparison of different materials for air-conditioning system mounting brackets has not been done.

2.3 Objectives of the Present Work

By considering the above mentioned gaps in the literature review, following objectives were identified:

(i) Linear static structural analysis to be performed.

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

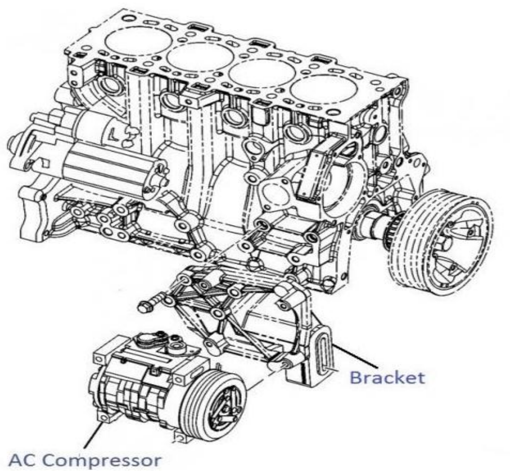
(iii) Optimization of bracket to be done with different techniques *i.e.*, topology and shape optimization to obtain maximum stiffness and minimum mass of a bracket.

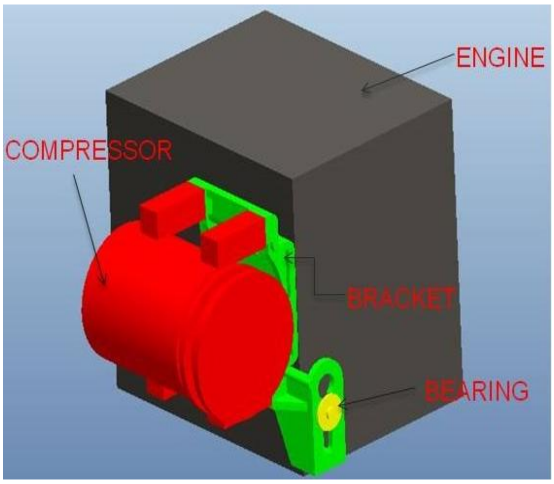
(iv) Comparison of initial bracket design and optimized design based on mass, stresses and displacements.

(v) Validation of results, both experimentally and numerically.

CHAPTER 3 ANALYSIS OF BRACKET

A bracket of the air conditioning system of an Indian passenger car was selected for analysis. The bracket to mount the compressor is fixed at the front end of the engine, on which the compressor is mounted. Figure 3.1 shows the location of bracket and compressor assembly in actual condition. The compressor is further connected with engine timing belt, which results in lot of vibrations and fatigue on the bracket. To present the failure of bracket, it is to be analysed.



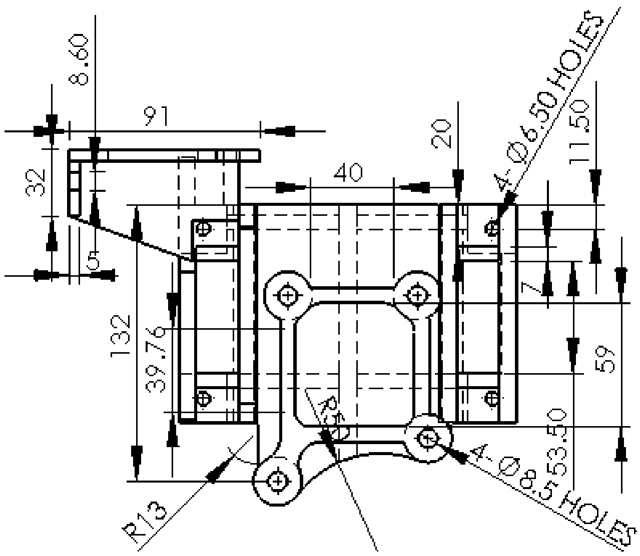


(a) **(b)**

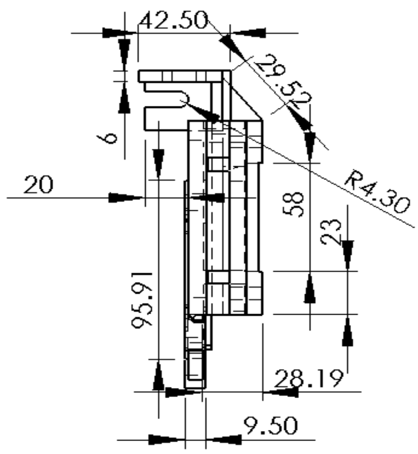
Figure 3.1: Location of compressor and bracket in car air conditioning system: (a) In real condition [40] ; (b) In CAD model

3.1 Reverse Engineering of the Bracket

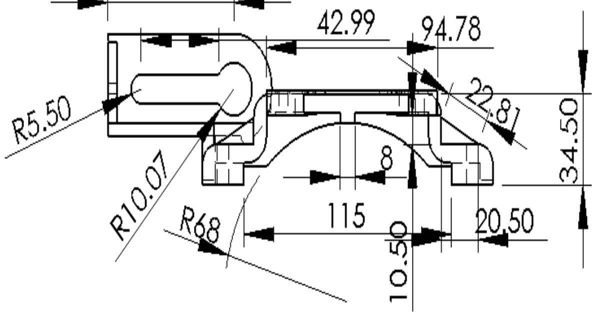
The drawing or CAD model of the selected bracket was not available for the present work. So, the bracket was purchased from the market and the reverse engineering of the bracket was done to make its drawings. Various metrological instruments *i.e.,* Coordinate Measuring Machine (CMM), Profile projector, Vernier calliper, Measuring scale, protector, radius gauge, depth gauge, divider, set scale, level meter, flat surface plate, etc. were used to accurately measure the various dimensions of the bracket. Different views of the bracket generated after reverse engineering are shown in Figure 3.2.



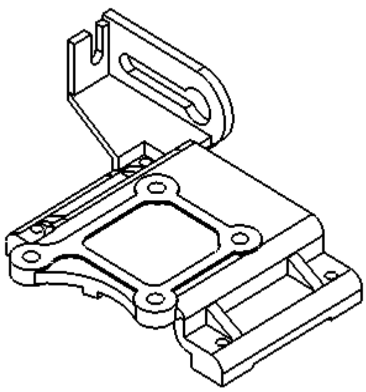
Front View



Left Side View



Top View



Isometric View

Figure 3.2: Dimensions of original Bracket (in mm)

3.2 Material Properties

The material of the bracket was analyzed using optical microscope. It was found to be Ductile Iron (Spheroidal Graphite Cast Iron). Its Scanning Electron Microscope (SEM) image is shown in Figure 3.3.

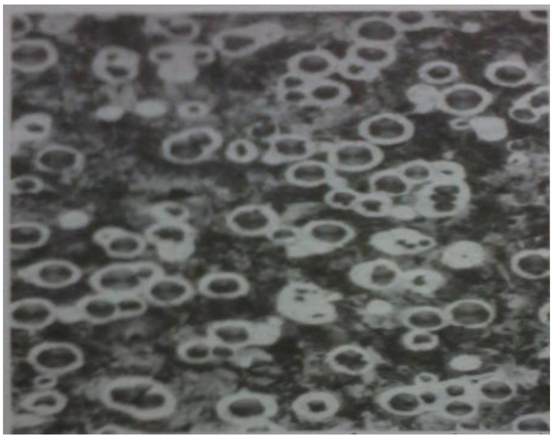


Figure 3.3: Microstructure of material of the bracket

Figure 3.3 shows bull's eye structure that involves Pearlite matrix with ferrite halos around graphite nodules. It was etched with 4% natal. It was concluded that the microstructure of a sample is Ferrite and Pearlite matrix., with the properties given in Table 3.1.

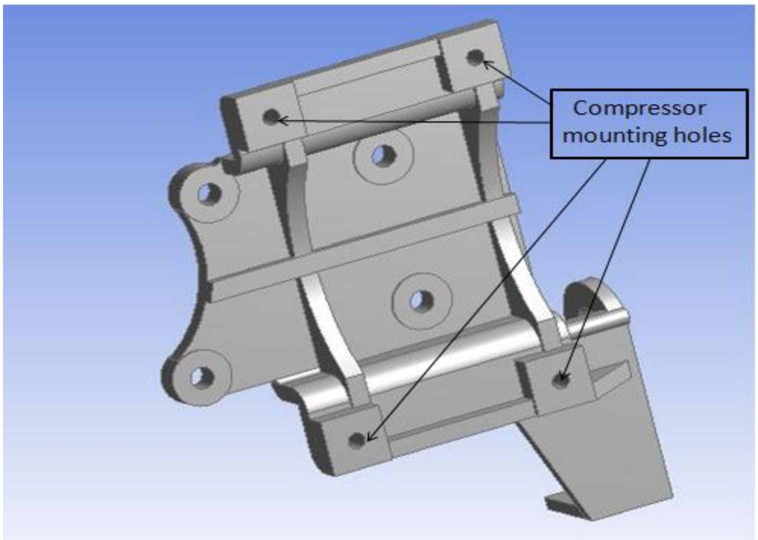
Table 3.1: Properties of the Material

Property	Value
Elastic Modulus	1.2e+011 N/m^2
Poisson's Ratio	0.31
Density	7100 kg/m^3
Tensile Strength	550 MPa
Yield Strength	350 MPa

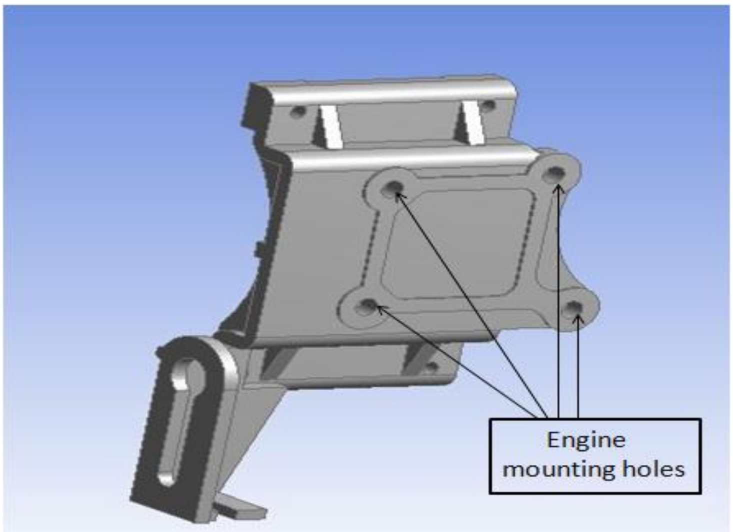
The properties in Table 3.1 are taken from ISO standard 1083(b) 550-5S **[41]** for Ductile Iron.

3.3 Modelling and Mesh Generation

Using the dimensions given in Figure 3.2, the solid model was generated in Pro-E solid modelling software. The generated model is shown in Figure 3.4.



(a)



(b)

Figure 3.4: Mounting Holes location for bracket:
(a) CAD model of bracket showing compressor mounting holes; (b) CAD model of bracket showing engine mounting holes

The file was converted into IGES format for mesh generation in ANSYS analysis software. The 3D tetrahedral elements were selected as the shape of the bracket is not uniform. The quality of tetrahedral element mesh generated was checked with respect to the following standard parameters **[1]**.

1. Aspect ratio

It is defined as the ratio of a maximum element edge length to the minimum element edge length.

For tetrahedral element, Ideal value = 1 (Acceptable <5)

2. Jacobian

It is a scale factor that comes into play due to the transformation from one coordinate system to another.

Ideal value = 1.0 (Acceptable > 0.5)

3. Volumetric Skewness

To calculate the value of volumetric skewness **[1]**, create a sphere that passes through the corner nodes of the tetrahedral element, fit an ideal tetrahedral in it. Then, find out the volume of ideal and actual tetrahedral elements.

Ideal value = 0 (Acceptable < 0.7)

$$volumetric\ skewness = \left[\frac{(V_{ideal} - V_{actual})}{V_{ideal}}\right] \tag{3.1}$$

4. Skewness

Skewness as shown in Figure 3.5, in tetrahedral element can be calculated by finding the minimum angle between the vector from each node to the mid points of the opposite edges.

Triangle skew = 90 – α

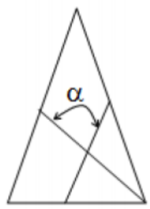


Figure 3.5: Skewness [1]

Figure 3.6 shows the mesh generated using tetrahedral element for the bracket. The value for the different quality parameters are given in Table 3.2.

Table 3.2: Mesh Quality check parameters values

Parameter	Value
Aspect ratio	2.22
Jacobian	1.04
Volumetric skewness	0.38
Skewness	45^0

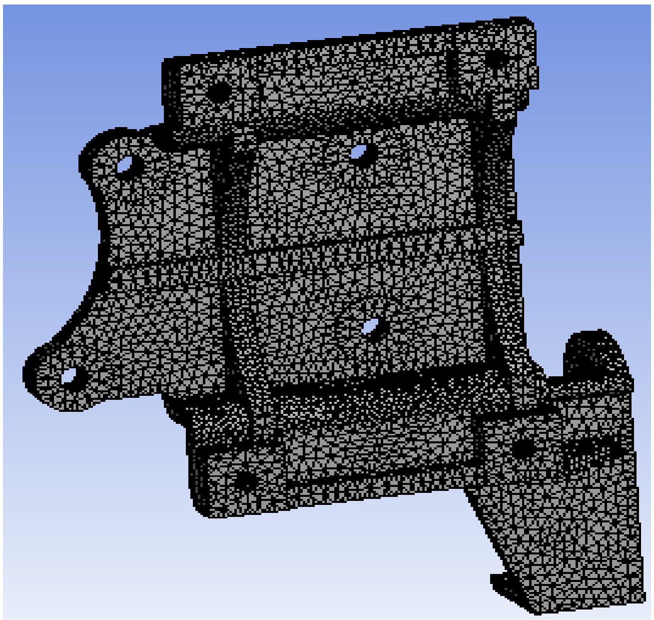


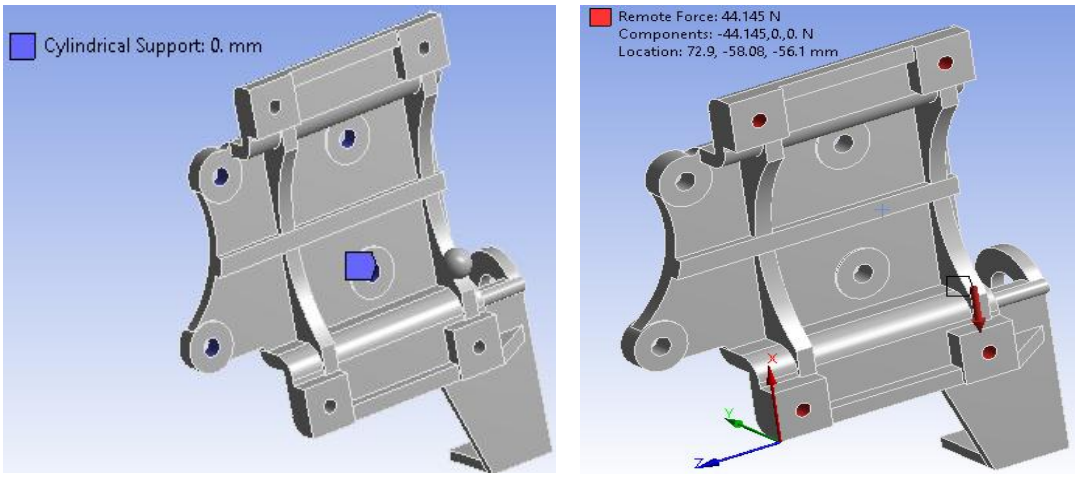
Figure 3.6: Meshed Model of the Bracket

3.4 Determination of Center of Mass

There was a need to determine the centre of mass of a compressor to find the location of application of remote force in static structural analysis and position of point mass in dynamic analysis of the bracket. For this, shape of a compressor with approximate dimensions was modelled. The position of centre of mass with respect to reference point was evaluated. The mass of a compressor was considered to be 4.5 kg, with the assigned density of material to be 2700 kg/m^3 **[42]**.

3.5 Static Structural Analysis

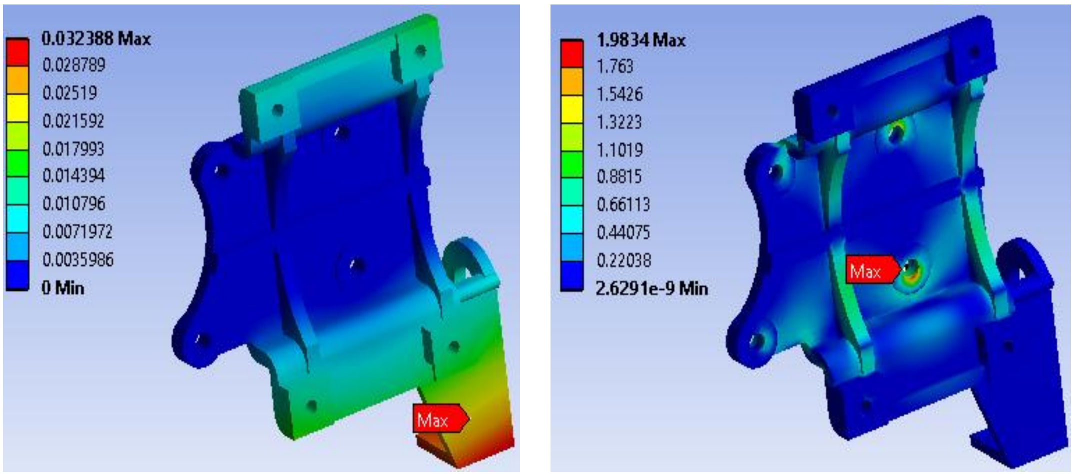
Static structural analysis was performed to obtain deformation and stresses in the bracket with the load equivalent to weight of the compressor applied on it. Linear static structural analysis was performed on the bracket in ANSYS software. Analysis was performed after applying the proper forces and boundary conditions on the meshed model of the bracket. The applied boundary conditions and forces are shown in Figure 3.7.



(a) (b)

Figure 3.7: Boundary conditions and Forces for static structural analysis: (a) Cylindrical support; (b) Remote force applied

The results obtained after analysis are shown in Figure 3.8.



(a) (b)

Figure 3.8: Results obtained after performing the static structural analysis: (a) Total Deformation (in mm); (b) Von mises stress (in MPa)

As can be seen from Figure 3.8, the selected bracket is over designed with higher factor of safety.

3.6 Dynamic Analysis

Dynamic analysis was performed to check the effect of vibration from engine to the compressor through the bracket. The following procedure was applied for dynamic analysis:

(i) Modal analysis or Normal Mode analysis was performed.

(ii) Modal frequency response analysis was performed.

The output of first analysis is the input parameters for the second type of analysis.

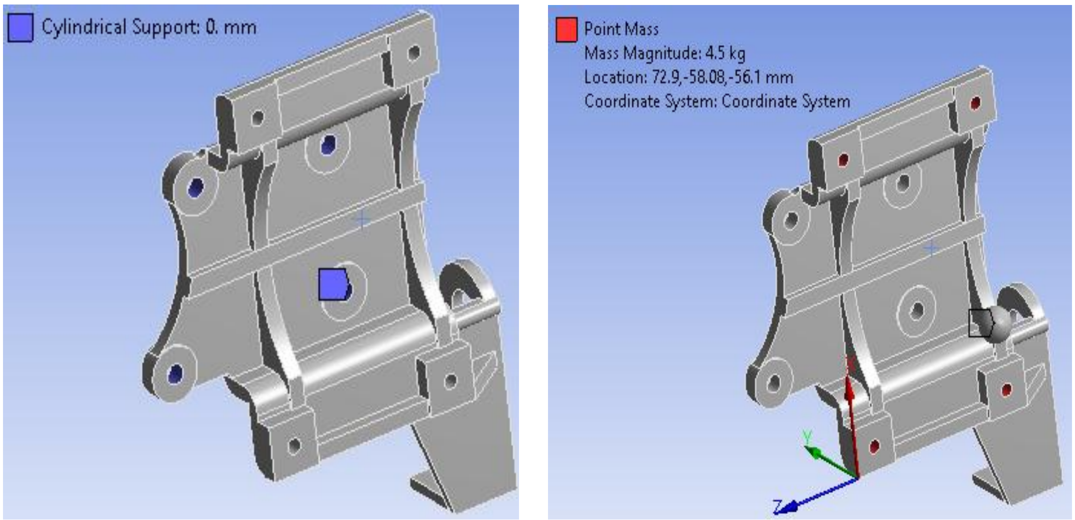
(i) Modal Analysis or Normal Mode Analysis

This type of analysis is basically performed to obtain the natural frequencies and their corresponding mode shapes. It is a first step towards dynamic analysis, output of which is further used in transient analysis, frequency response analysis and random response analysis.

The following parameters were considered for the Modal analysis:

- **(a) External force:** No external force was applied, as per standards, to find natural frequencies.
- **(b) Constraints:** In the present work, the natural frequencies were found under real conditions.
- **(c) Damping:** Damping is neglected to find the natural frequency.
- **(d) Analysis output:** The results obtained through this analysis are magnitude of frequencies and mode shapes.

The modal analysis of the bracket was performed in ANSYS after mesh generation using tetrahedral elements. The boundary conditions were applied as per Figure 3.9(a). The load of 4.5 kg was applied at the center of mass of the compressor, shown in Figure 3.9(b).



(a) **(b)**

Figure 3.9: Boundary conditions for modal analysis as well as frequency response analysis: (a) Cylindrical support; (b) Point mass that acts as a compressor

The natural frequencies and their corresponding mode shapes obtained after the modal analysis are shown in Figure 3.10.

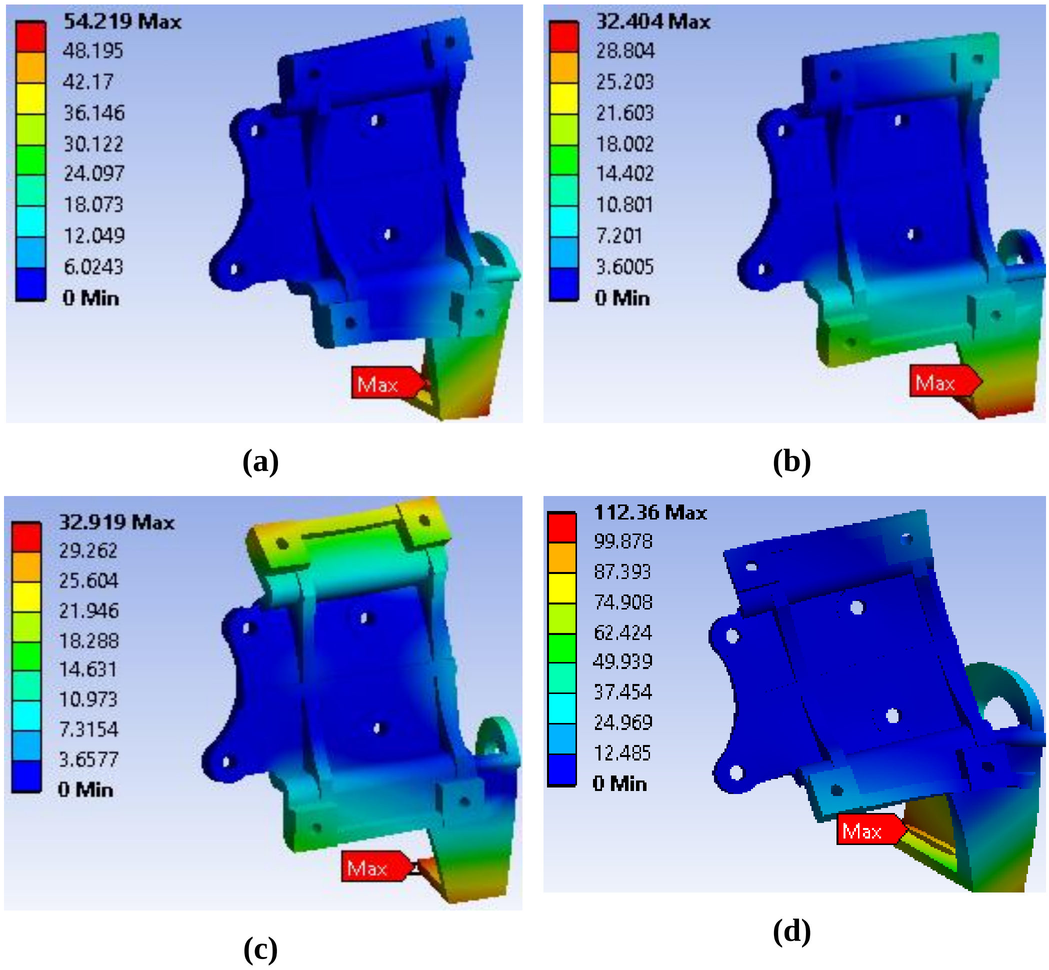


Figure 3.10: Mode Shapes for the bracket: (a) Mode shape 1 at 99.746 Hz; (b) Mode shape 2 at 116.21 Hz; (c) Mode shape 3 at 186.36 Hz; (d) Mode shape 4 at 246.82 Hz

The mode shape corresponds to the different patterns of vibration that is executed by bracket at a specified frequency. It corresponds to deformation that occurs in the bracket at its natural frequencies. The first natural frequency of the bracket is 99.746 Hz and corresponding to this frequency the deformation is under permissible limit.

(ii) Modal Frequency Response Analysis of Compressor Mounting Bracket

Modal frequency response analysis is performed on the bracket after exciting it by harmonic response in the form of $F_0 \sin\omega t$, where ω is the frequency of the excitation. During resonance, this frequency becomes equal to one of the natural frequency of the component that causes a lot of vibrations and stresses in the component. The analysis is performed in x, y and z direction which corresponds to the direction of excitation force. The analysis takes input parameters from modal analysis and results are the stresses at various frequencies.

a) Modal frequency response analysis along x-axis

The excitation force was applied along x-direction as per Industrial Standards (JIS1601D) **[44]**. As per the standard, different procedure is given for different components as per their location and mounting. In the selected bracket, type D was chosen which is used for the components attached directly to the engine. For the analysis, the boundary conditions and loads are shown in Figure 3.9. An Inertial load of acceleration of 30g *i.e.,* 294300 mm/sec^2 was applied along x-direction for excitation of the base of bracket. The applied acceleration is shown in Figure 3.11.

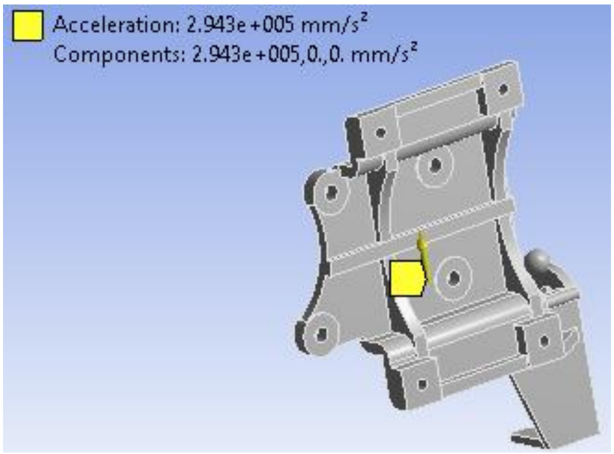


Figure 3.11: Direction of acceleration applied in x axis

Frequency range was considered to be from 90 Hz to 400 Hz with 50 numbers of intervals. The damping ratio of 6% was considered as per the standards. The damping ratio is the ratio of actual damping coefficient to the critical damping coefficient.

The results after the analysis shows the von-mises stresses produced along x-axis due to dynamic loading with selected frequency of 99.746 Hz. Figure 3.12 shows that the stresses in the bracket are low as compared to the yield strength of the material.

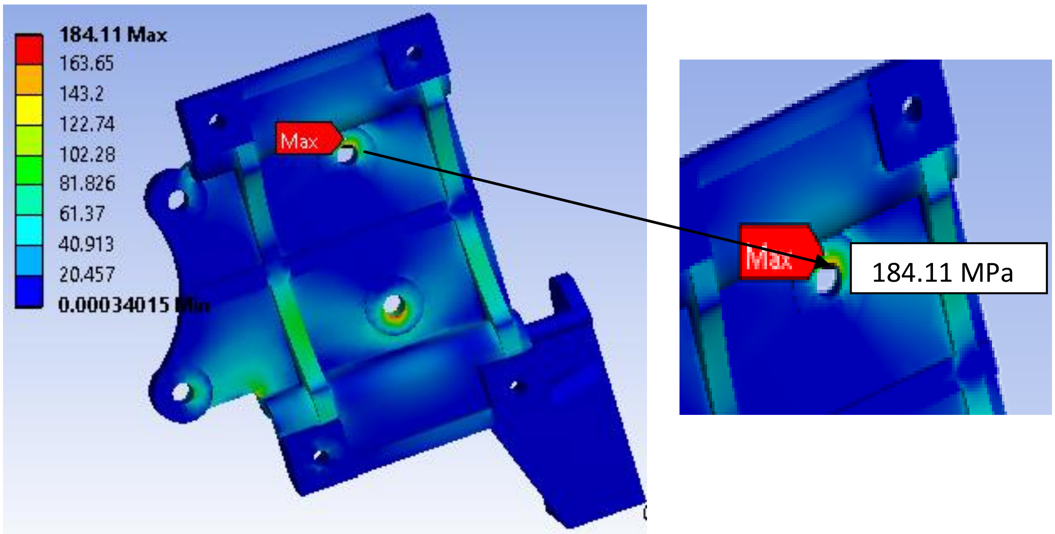


Figure 3.12: Von mises stresses produced in x-axis at frequency 99.746 Hz.

b) Modal frequency response analysis along y-axis

After applying the acceleration along y-axis, the von-mises stresses produced at the frequency of 99.746 Hz are shown in Figure 3.13.

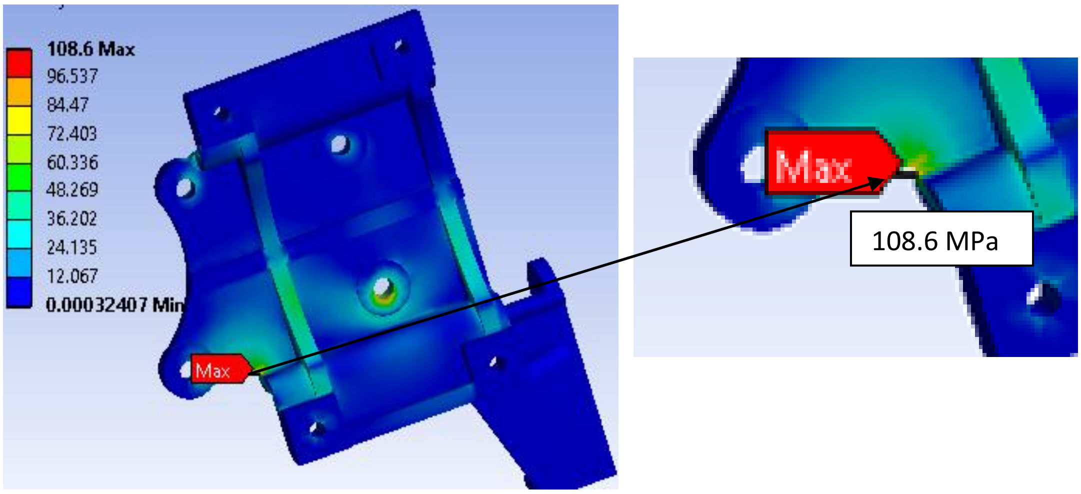


Figure 3.13: Von mises stresses produced in y-axis at frequency of 99.746 Hz.

c) Modal frequency response analysis along z-axis

After applying the acceleration along z-axis, the von-mises stresses produced at the frequency of 99.746 Hz are shown in Figure 3.14.

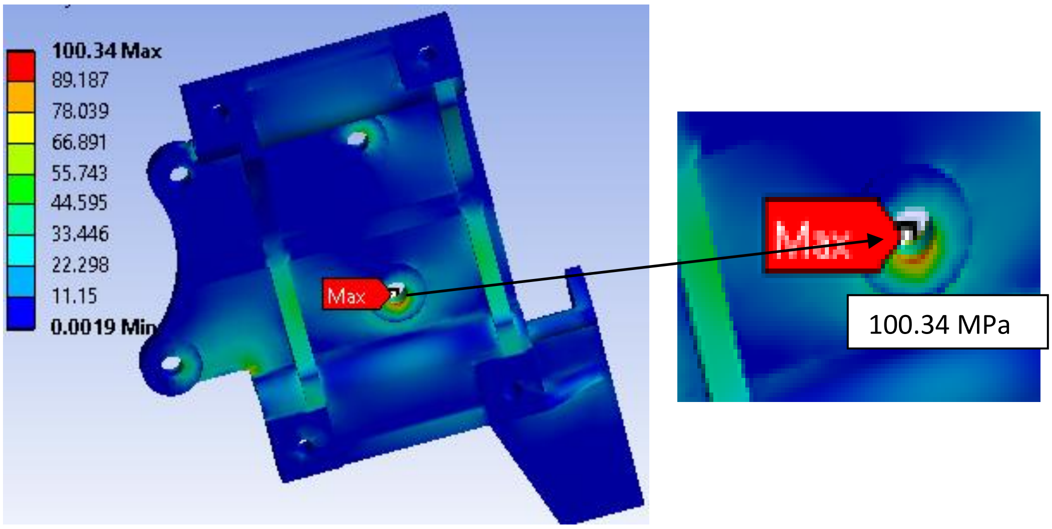


Figure 3.14: Von mises stresses produced in z-axis at frequency of 99.746 Hz

As can be seen from Figure 3.12 to 3.14, the design of the selected bracket is safe for dynamic loading conditions.

CHAPTER 4 OPTIMIZATION OF BRACKET

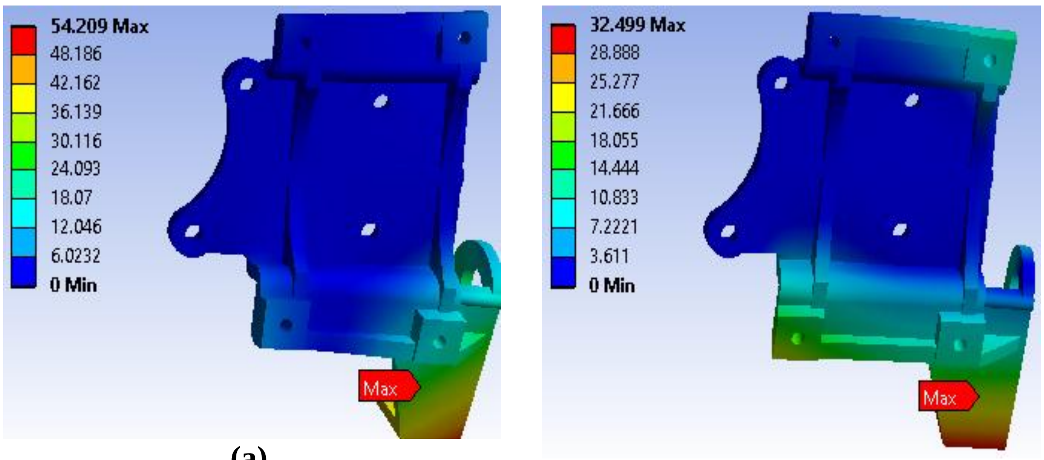
4.1 Shape Optimization of Original Bracket Design

Shape optimization is a method of optimization in which the objectives defined for optimization are lesser in number as compared to constraints. It includes fillet radius, hole diameter, thickness, etc. as design variables. The optimization was done in ANSYS after doing the structural and dynamic analysis on the bracket. Response surface optimization was used with mass and frequency as the objective parameter. The objective function and constraints used in optimization are shown in Figure 4.1.

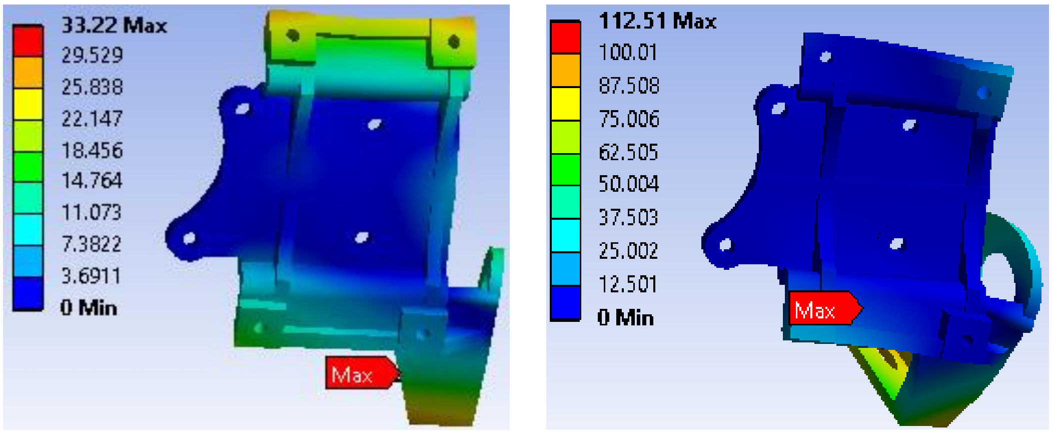
	A	B	C	D	E	F
1	Name	Parameter	Objective		Constraint	
2			Type	Target	Type	Lower Bound
3	Minimize P3	P3 - Solid Mass	Minimize		No Constraint	
4	Maximize P2; P2 >= 100 Hz	P2 - Total Deformation Reported Frequency	Maximize		Values >= Lower Bound	100

Figure 4.1: Objective function and constraints used in shape optimization

After the optimization process, the changes were made in the geometry. Figure 4.2 shows the natural frequencies corresponding to their mode shapes after shape optimization of the original bracket design.



(a) **(b)**



(c) (d)

Figure 4.2: Total deformation (in mm) at different natural frequencies:
(a) Mode shape 1 at 100.72 Hz; (b) Mode shape 2 at 116.81 Hz;
(c) Mode shape 3 at 185.91 Hz; (d) Mode shape 4 at 248.02 Hz

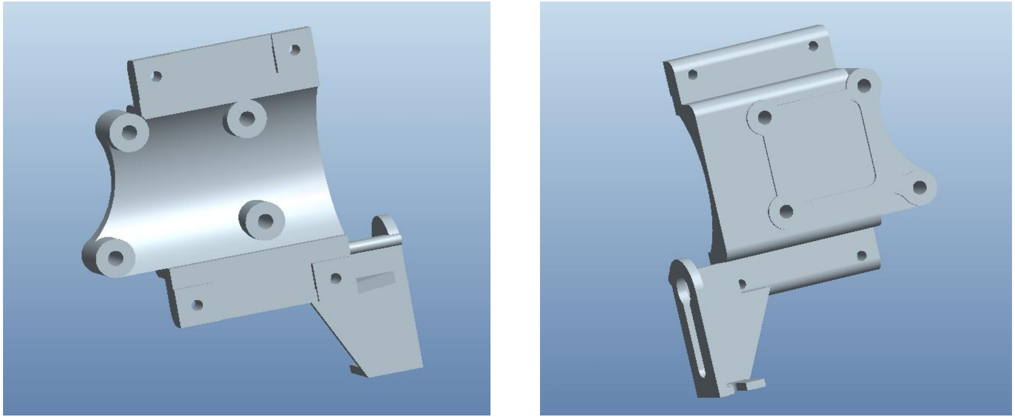
The mass of the bracket obtained after performing the shape optimization was 1.627 kg. As can be seen from Figure 4.2, there is minor change in natural frequencies and mass for optimized bracket as compared to the original bracket. Thereafter, topology optimization was performed on the bracket for better results.

4.2 Topology Optimization of Original Bracket Design

It is a mathematical approach to optimize the layout of material that is available within the defined design space. It also includes various constraints and loads applied on a part. The result after performing this optimization satisfies all the given boundary conditions. It is generally used at the conceptual stage in designing a product, to reduce the design development iterations. Sometimes, this optimization gives a shape that is unable to manufacture. To handle these types of challenges, manufacturing constraints are used effectively for a product. The following steps were used for the topology optimization of the bracket.

4.2.1 Base Design for Topology Optimization

The final result of topology optimization depends upon the shape and mass of base design used for the analysis. So, the design space plays a major role in topology optimization. In the present work, the base design was modelled according to the shape of final bracket required. Figure 4.3 shows the base design that was used for the optimization. The mass of the base design for Spheroidal Graphite Iron (SGI) was 2.35 kg.



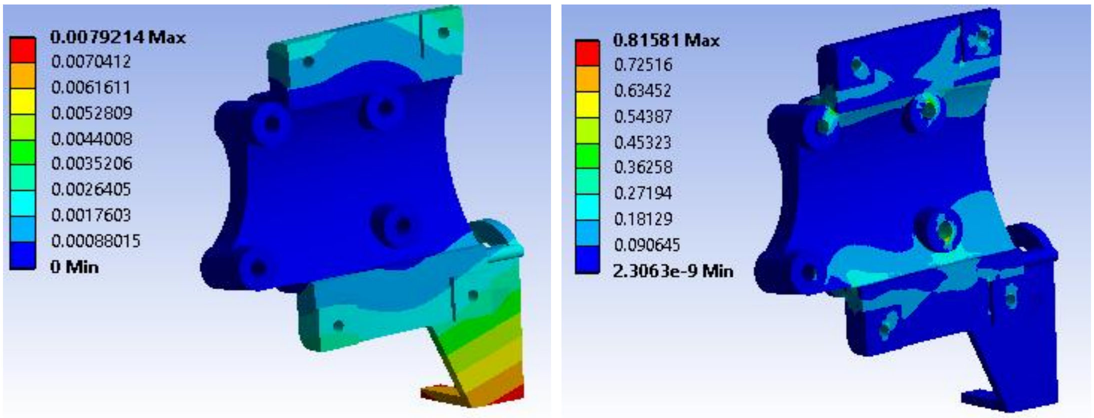
(a) (b)

Figure 4.3: Views of Base Design of Bracket for Topology Optimization:

(a) Front View; (b) Rear View

4.2.1.1 Linear static structural analysis

The linear static structural analysis was performed on the base design of bracket as per the discussion in the section 3.5. Figure 4.4 shows the deformation and von-mises stresses in the bracket.



(a) (b)

Figure 4.4: Results obtained after static structural analysis of base design:

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

It can be seen from the Figure 4.4 that the maximum stress in the bracket is below the yield point of the material. So, the design is having the higher factor of safety.

4.2.1.2 Modal analysis

Modal analysis was performed on the bracket as per discussion in the section 3.6. The design and non design space for the bracket is shown in Figure 4.5.

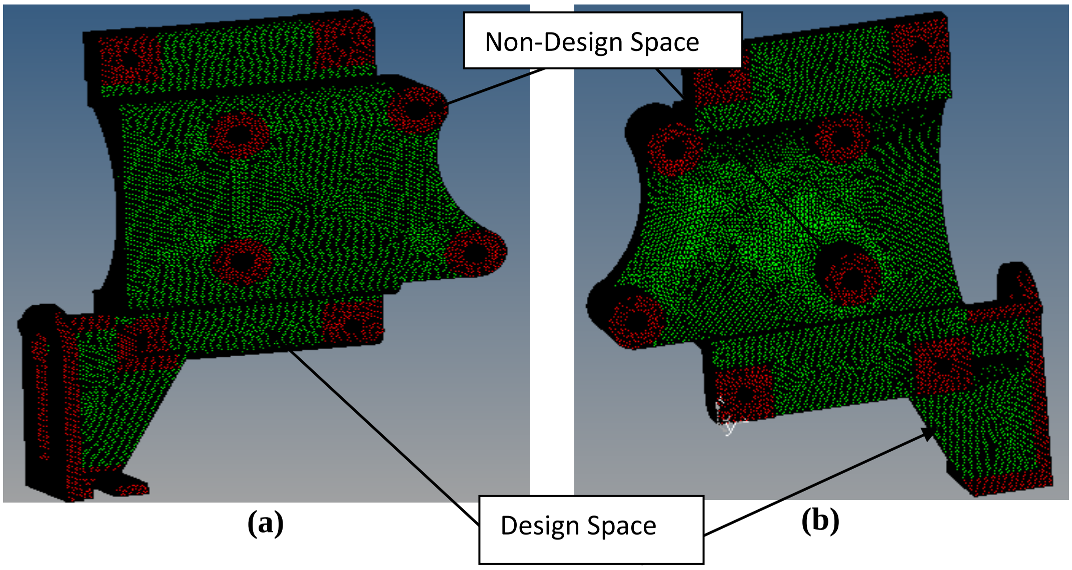


Figure 4.5: Design and non- design space for topology optimization

(a) Front view; (b) Back view

For the topology optimization, the concept of Rigid Body Element (RBE) was used to define the compressor mass and bolt locations that act as the boundary conditions for the problem.

RBE2: This element was used to define bolt location. It creates a rigid region by generating equations of constraints to relate the nodes in a region. Geometric location must be assigned to the nodes that are in the rigid region. It is used to translate degree of freedom at the dependent nodes. It requires two types of nodes:

Master: It is a node which represents the fixture. It is an independent node and was created at the position of fixture mounting as shown in Figure 4.6.

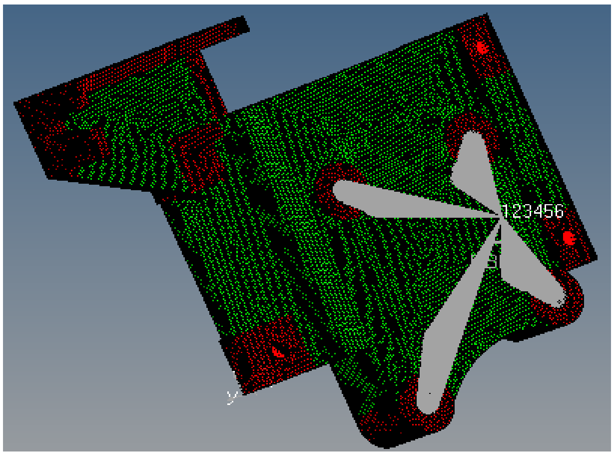


Figure 4.6: Master node fixed at position of fixture

Slave: Other selected nodes except master node are on mounting bracket (bolt location) represent the “slave nodes”. These nodes are dependent nodes.

RBE3: This element also creates a rigid region. In this element, master node is a dependent node and slave nodes are independent nodes. Here, in this problem it represents the mass of compressor at master node, which is created at the centre of mass of compressor. Figure 4.7 shows the concept of RBE3 element.

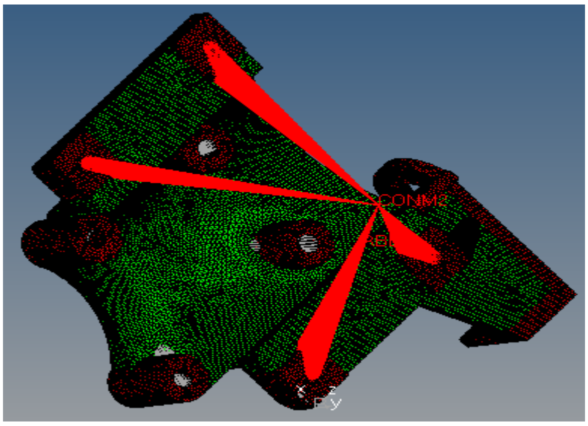


Figure 4.7: RBE 3 element with CONM2 mass of 4.5 kg

It is used to distribute the effect of a mass (that is acting at the dependent node) over a number of other nodes (that are the independent nodes), which doesn’t add stiffness to the structure.

The boundary conditions and the load steps used to perform modal analysis are given below:

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

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

b) Constraints (load collector): In this load collector, node1 is fixed along all directions as shown in Figure 4.8.

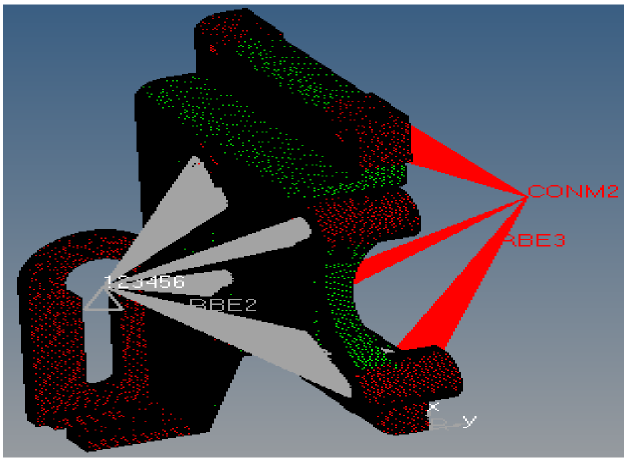


Figure 4.8: Boundary conditions used in modal analysis

(ii) After this, load step was defined for modal analysis or normal modes.

(iii) After modal analysis, natural frequencies of a base design for topology optimization are shown in Table 4.1.

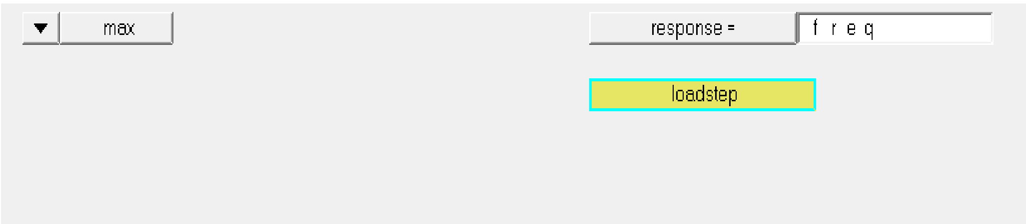
Table 4.1: Natural frequencies of bracket corresponding to its modes

Mode	Natural frequency(Hz)
1	186.26
2	256.10
3	316.25
4	339.25
5	407.57
6	576.46

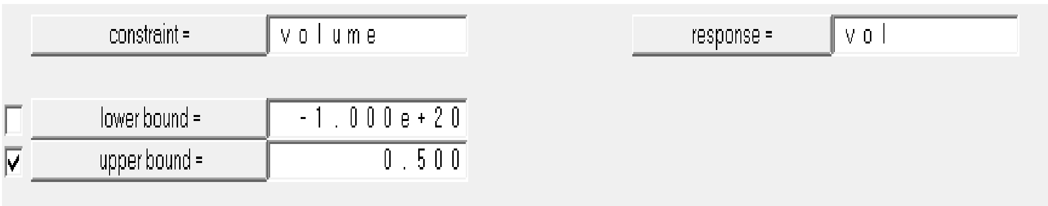
4.2.2 Topology Optimization

After structural and modal analysis, the topology optimization was done. It optimizes the material layout of a design space defined in the component with respect to the defined objective function and constraints. The design and non-design space for this optimization is already defined in section 4.2.1.2. The load step used for optimization was modal analysis or normal modes in Hypermesh software.

The objective function and constraints were defined as shown in Figure 4.9. As can be seen from the figure, natural frequency as objective function with the volume fraction of 0.5 as constraint was selected for optimization.



(a)



(b)

Figure 4.9: Objective function and constraints defined for topology optimization: (a) Objective function; (b) Constraints

The manufacturing constraint *i.e.*, draw direction was also defined for easier manufacturability of bracket. After solving, the element density plot was generated with respect to the defined constraints and objective function. Figure 4.10 shows the element density after performing the topology optimization. It can be seen from the Figure 4.10 that the material can be removed up to element density of 0.5.

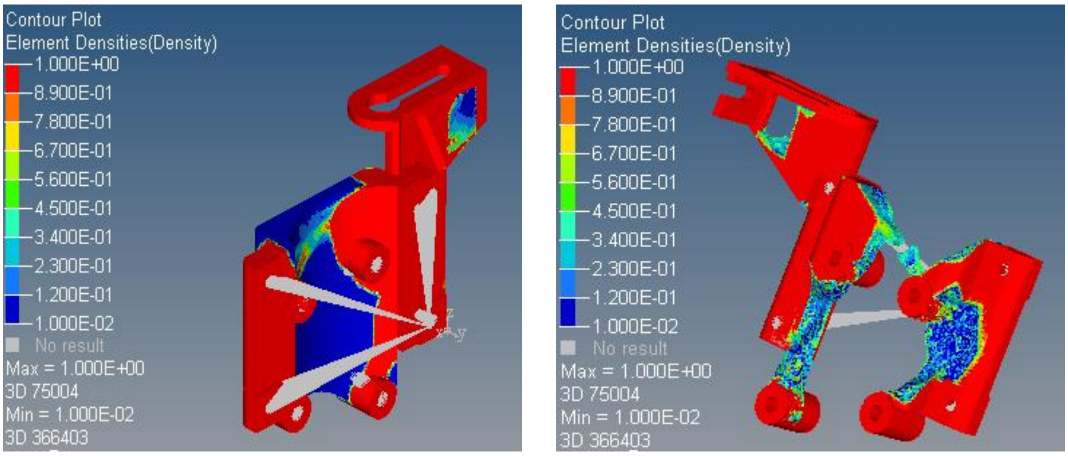


Figure 4.10: Element density after performing topology optimization

Figure 4.11 shows the design of the bracket after topology optimization.

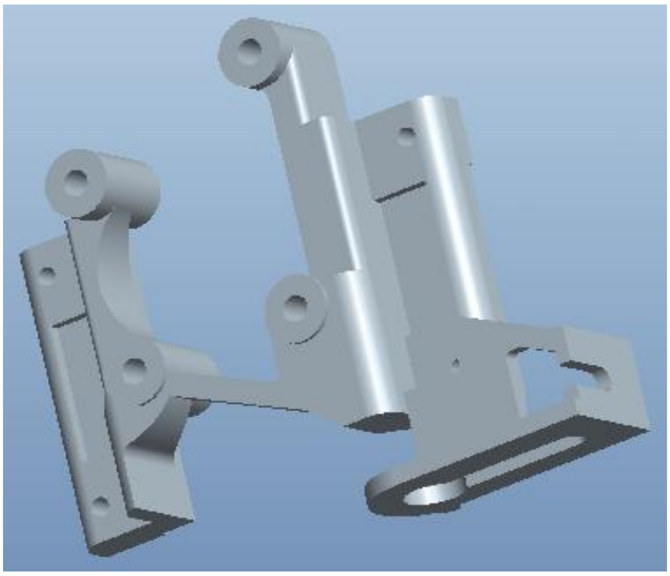


Figure 4.11: Design of bracket after topology optimization

4.3 Analysis of Bracket after Topology Optimization

Dynamic analysis was done on the optimized design of the bracket as per the procedure discussed in previous chapter. Figure 4.12 shows the mode shapes and deformation corresponding to different natural frequencies.

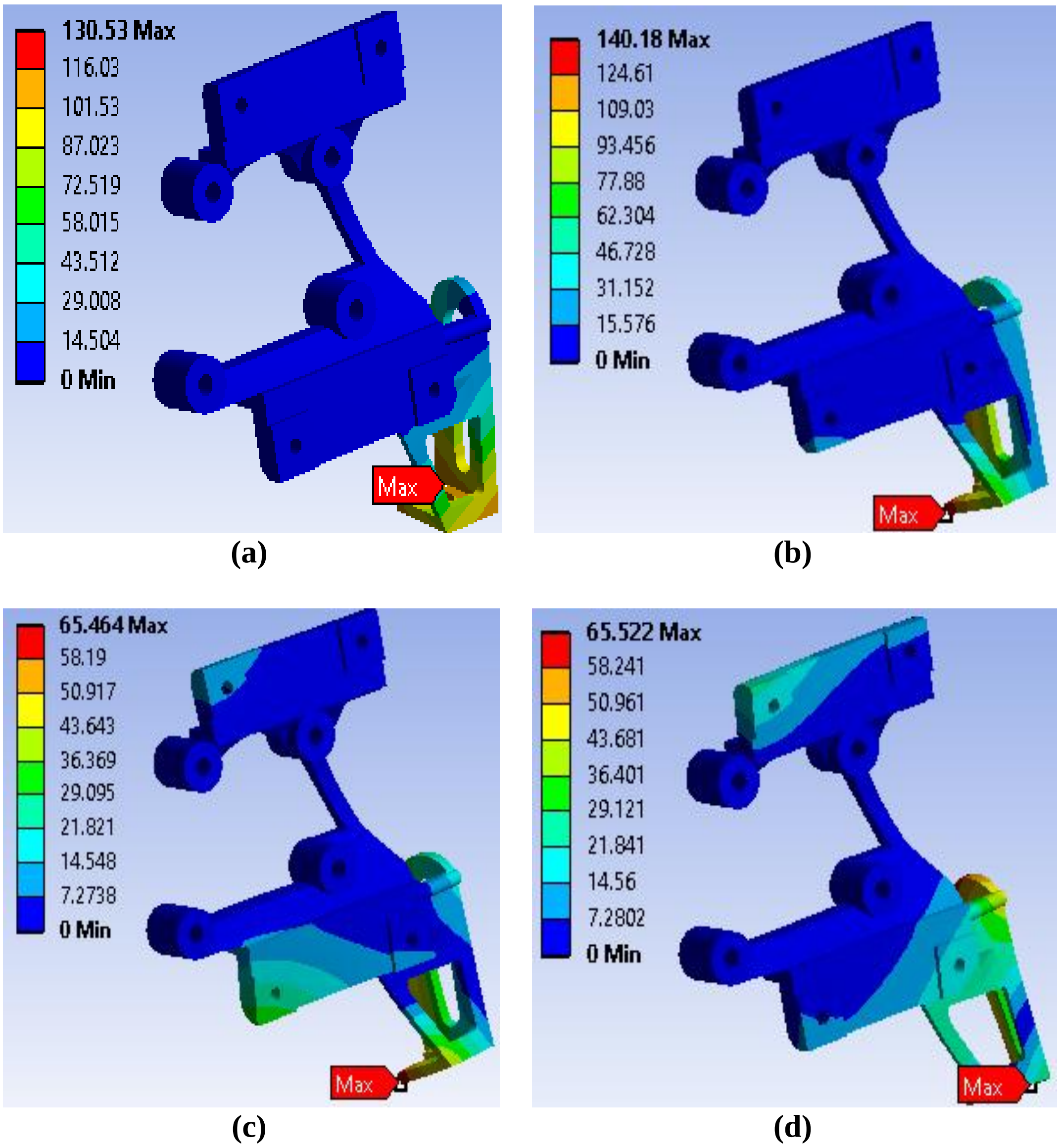


Figure 4.12: Total deformation (in mm) at different natural frequencies

(a) Mode shape 1 at 158.81 Hz; (b) Mode shape 2 at 230.16 Hz;
(c) Mode shape 3 at 265.85 Hz; (d) Mode shape 4 at 345.79 Hz

It can be seen from the Figure 4.12 that the natural frequency of the bracket has been improved as compared to the original design. This means that the stiffness of the bracket has increased.

Thereafter, the frequency response analysis was performed in x, y and z direction as per the discussion done in section 3.6. Figure 4.13 shows the stresses at the first natural frequency.

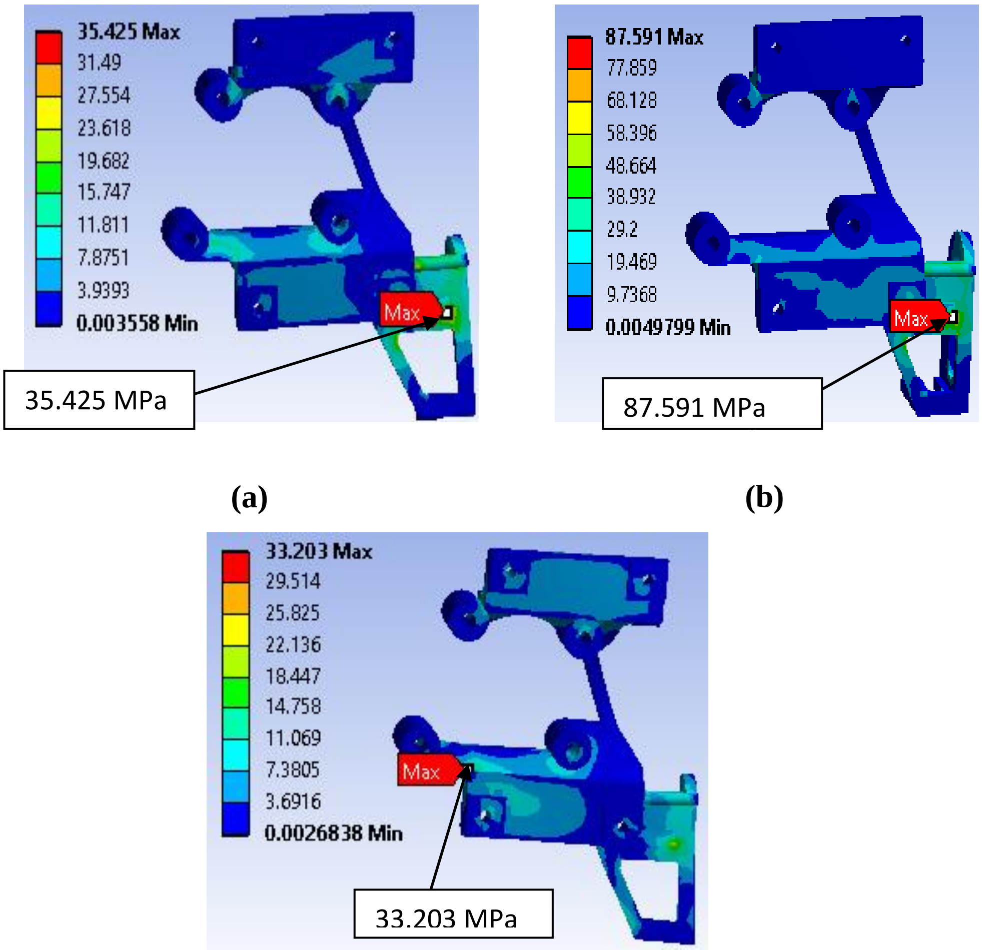


Figure 4.13: Von- Mises stresses at First natural frequency:

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

It can be seen from Figure 4.13 that the stresses produced in the optimized bracket are well within the prescribed limit. The mass of the bracket was reduced to 1.60 kg with the first natural frequency of 158.81 Hz.

4.4 Shape Optimization after Topology Optimization

Shape optimization was also performed on the optimized design of the bracket. The objective function and constraints selected for the shape optimization are shown in Figure 4.14.

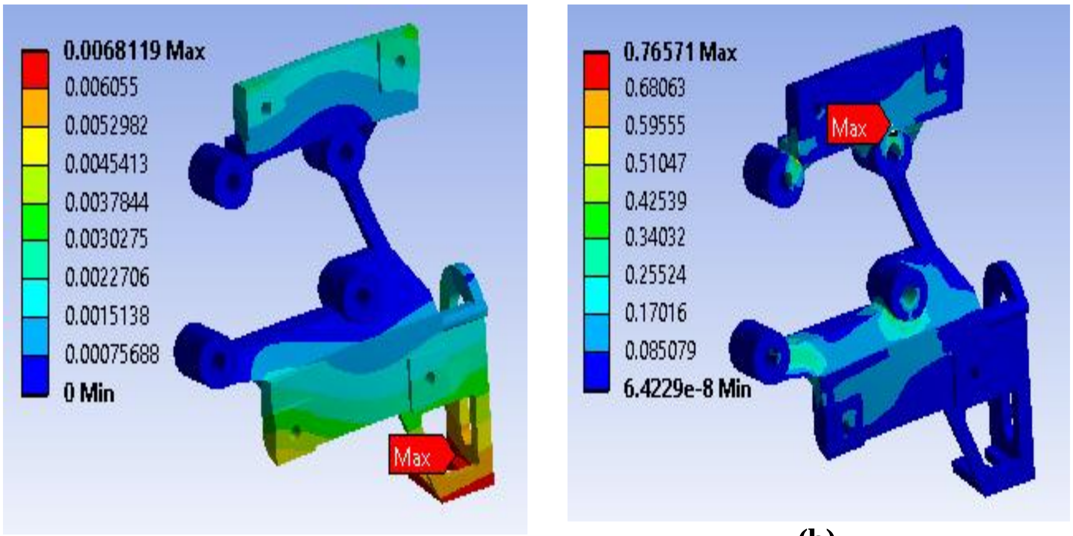
1	Name	Parameter	Objective		Constraint	
2			Type	Target	Type	Lower Bound
3	Minimize P7	P7 - Solid Mass	Minimize		No Constraint	
4	P8 >= 168 Hz	P8 - Total Deformation Reported Frequency	No Objective		Values >= Lower Bound	168

Figure 4.14: Objective function and constraints used in shape optimization

The design of bracket obtained, on performing shape optimization is almost the same as obtained after topology optimization. There were minor variations only in the dimensions that were defined in the shape optimization as design variables.

4.5 Static and Dynamic Analysis after Shape and Topology Optimization

Structural and dynamic analysis of a bracket after shape and topology optimization was performed as per the procedure discussed in section 3.5 and 3.6. Figure 4.15 shows the deformation and stresses after static structural analysis of the design of the bracket after shape optimization and topology optimization. Figure 4.16 shows the deformation of the bracket at different natural frequencies.



(a) **(b)**

Figure 4.15: Results after Static Structural Analysis:

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

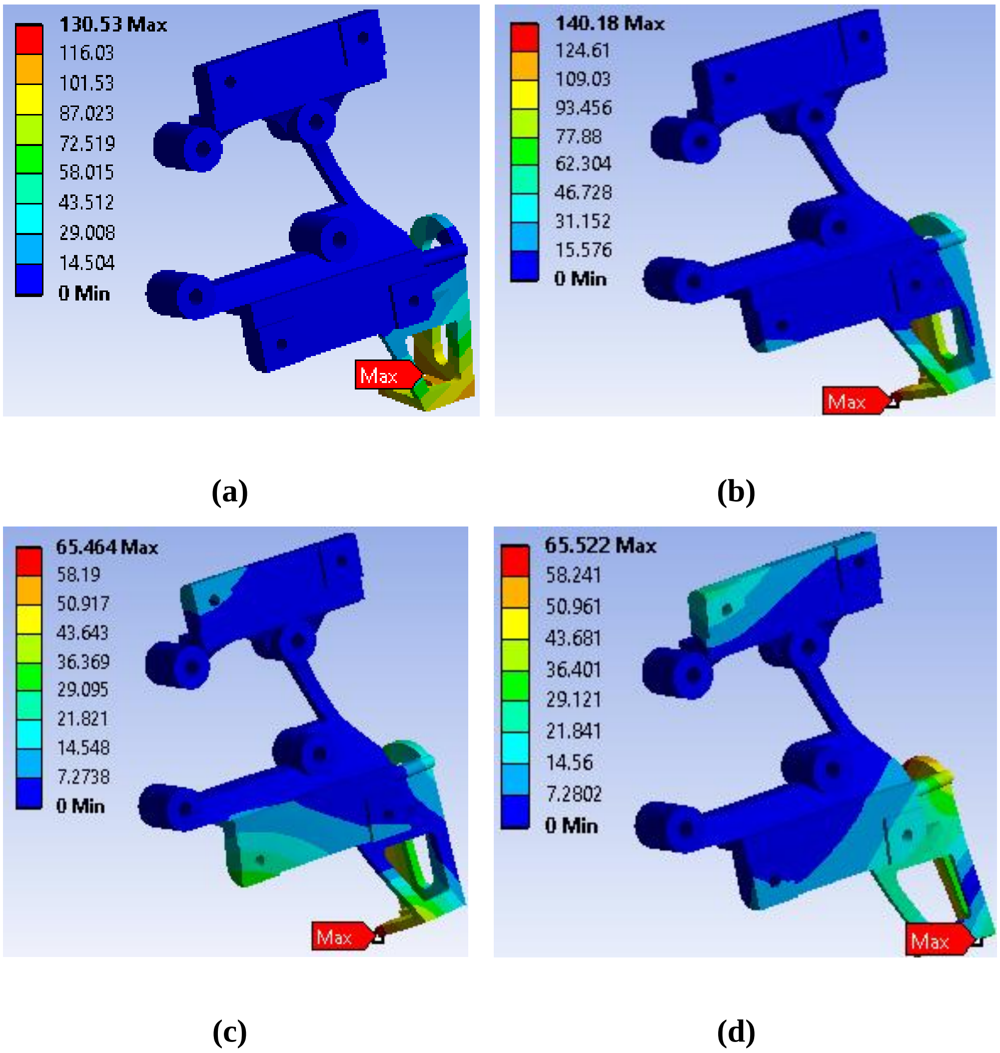


Figure 4.16: Total deformation (in mm) at different natural frequencies

(a) Mode shape 1 at 168.69 Hz ; (b) Mode shape 2 at 235.99 Hz ;
(c) Mode shape 3 at 265.67 Hz; (d) Mode shape 4 at 343.49 Hz

It can be seen from the Figure 4.16 that the total deformation about mean position of bracket at it natural frequency is under permissible limits. The natural frequency of the bracket is also more than the initial designs. This means new design of the bracket is more stiffen and isolated from vibration. Figure 4.17 shows the results obtained after performing frequency response analysis on optimized bracket *i.e.*, shape and topology optimization by applying excitation acceleration of 30g in x, y and z direction.

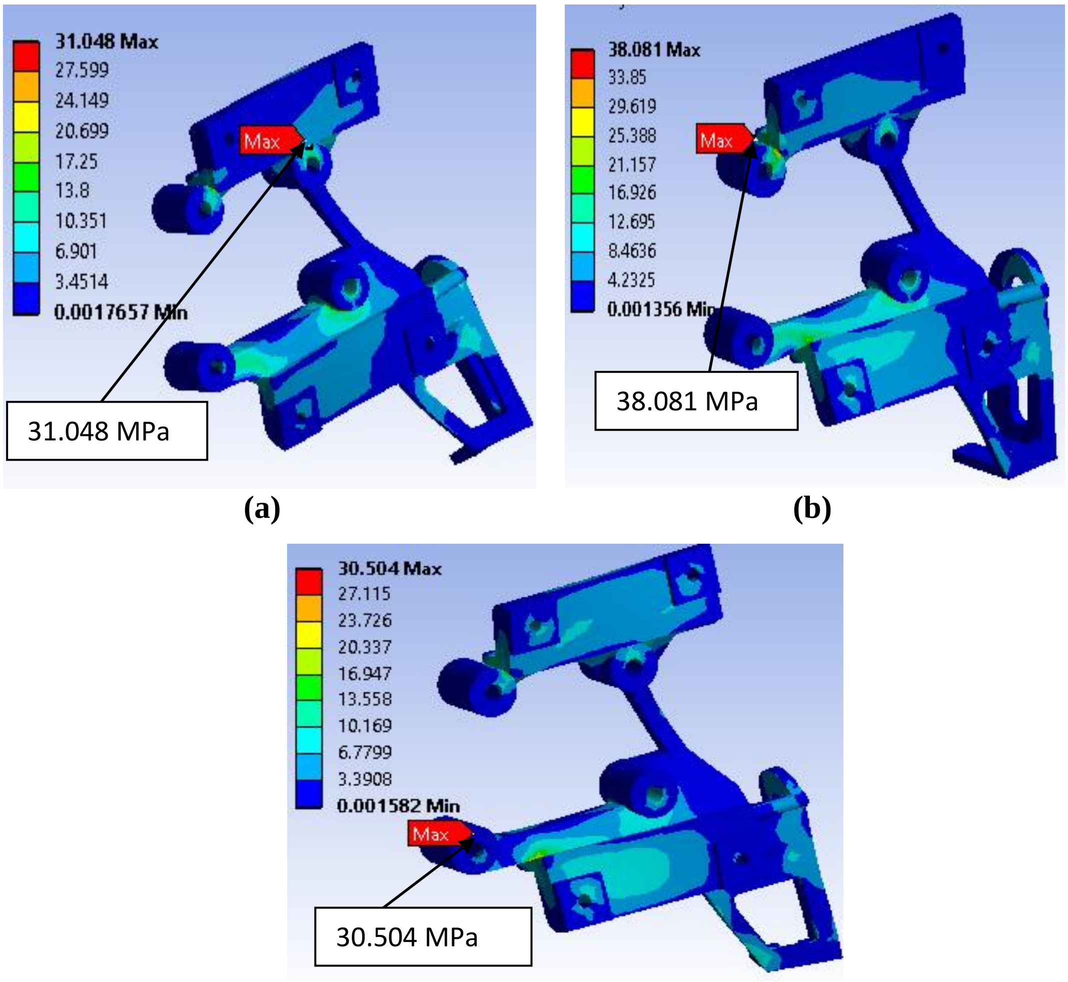


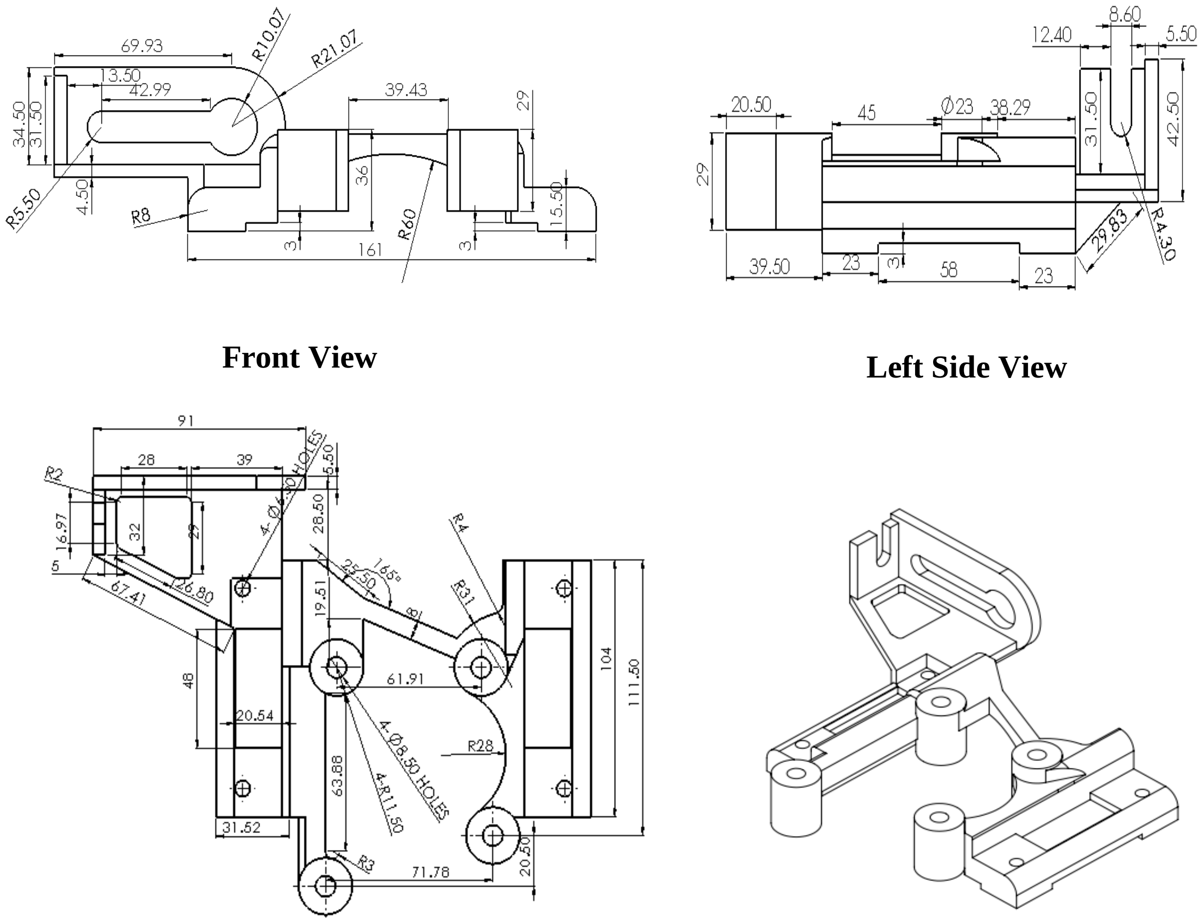
Figure 4.17: Von-Mises stresses at First natural frequency:

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

It can be seen from the Figure 4.17 that the von-mises stresses produced in the optimized bracket are under the permissible limit at its first natural frequency. So, the design of optimized bracket is safe under both static and dynamic conditions.

4.6 Optimized Design of Bracket

The final optimized design after both topology and shape optimization is shown in Figure 4.18. The reduction of mass with increase in natural frequency was observed in the optimized design. Mass and natural frequencies of the optimized design of the bracket are given in Table 4.2.



Front View

Left Side View

Top View

Isometric View

Figure 4.18: Dimensions of optimized design of bracket (in mm)

Table 4.2: Mass and Natural frequencies of Final Design of Bracket

Parameters	Value
Mass of final design of bracket	1.590 kg
1st natural frequency	168.69 Hz
2nd natural frequency	235.99 Hz
3rd natural frequency	265.67 Hz
4th natural frequency	343.49 Hz

From Table 4.2, it can be seen that the mass of optimized bracket is 1.590 kg with the first natural frequency of 168.69 Hz which shows that it is more stiffen than the original bracket design.

CHAPTER 5 EXPERIMENTAL VALIDATION

5.1 Experimentation

The optimized design of bracket obtained through CAE tools was validated experimentally. The bracket was casted and tested on the test rig.

5.1.1 Casting of optimized bracket

Firstly, the pattern was made of wood as per the optimized design of the bracket. Due allowances were given to compensate the change in dimension during solidification. Figure 5.1 shows the view of the prepared pattern.

Figure 5.1: Pattern for optimized bracket

After the casting of the bracket, shown in Figure 5.2, the holes were drilled. Thereafter the finishing of the bracket was done using milling and grinding processes. The final casting of the bracket is shown in Figure 5.3.

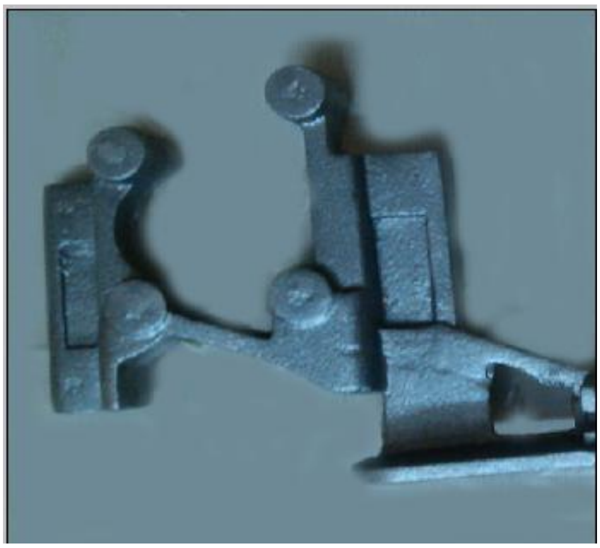


Figure 5.2: Casting of optimized bracket

Figure 5.3: Optimized bracket after machining

5.1.2 Fixture Design

A Fixture was designed to mount original and optimized bracket on electrodynamics vibration shaker. Electrodynamics vibration shaker is used to measure the response of a component upon excitation in the form of sine function or random function. In the present work, it was used to measure the natural frequencies under constraint conditions of brackets. Material of the fixture was chosen such that its natural frequencies do not match with any of the natural frequency of the bracket. So, mild steel was chosen as the material for fixture with the properties given in Table 5.1.

Table 5.1: Properties of mild steel used for Fixture [43]

Properties	Value
Young's modulus (N/m^2)	2e11
Poisson's ratio	0.3
Density(kg/m^3)	7860

After choosing the material for the fixture, the design of the fixture was made taking the reference of the shaker table of the machine. After a few iterations, the thickness of the fixture plate was finalized at 15mm with the total mass of the fixture to be 23 kg. The final design of the fixture is shown in Figure 5.4.

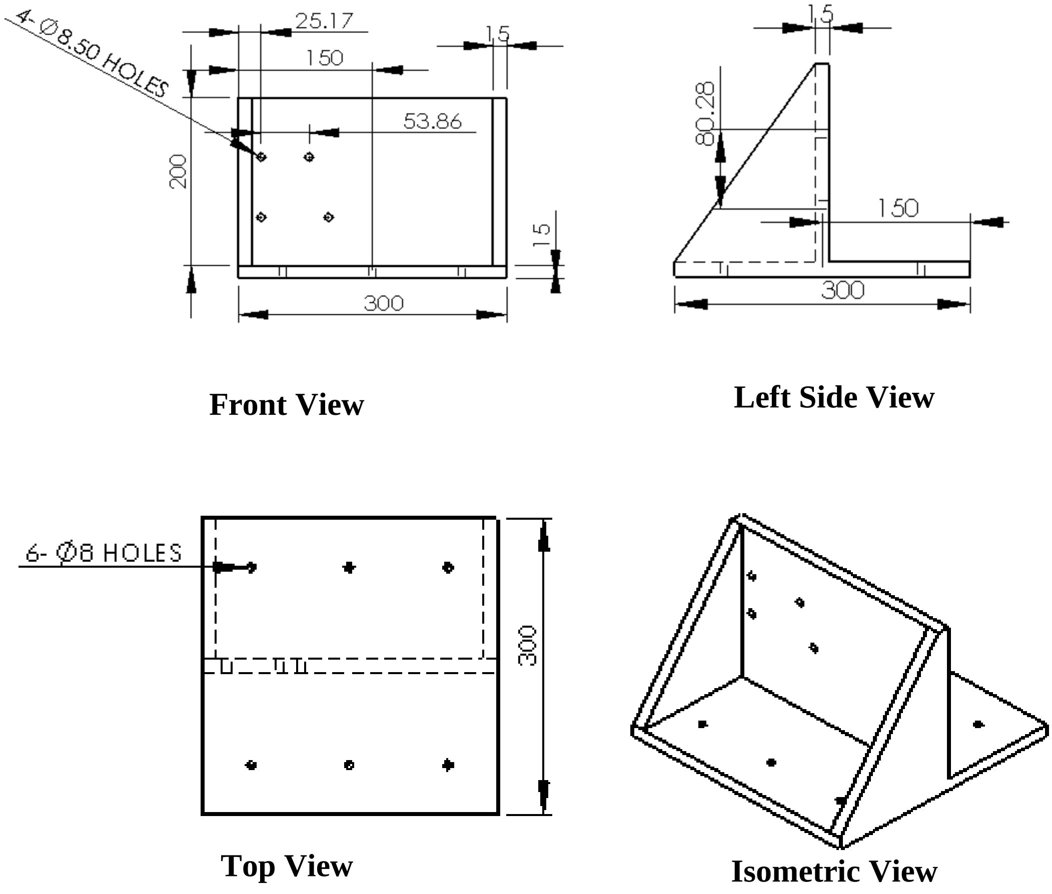


Figure 5.4: Dimensions of fixture (in mm)

The bracket fixed in the designed fixture on the machine is shown in Figure 5.5.

Figure 5.5: Position of bracket on fixture

5.1.3 Experimental Setup for Modal Testing

Figure 5.6 shows the experimental setup for finding the natural frequencies of brackets under different conditions.

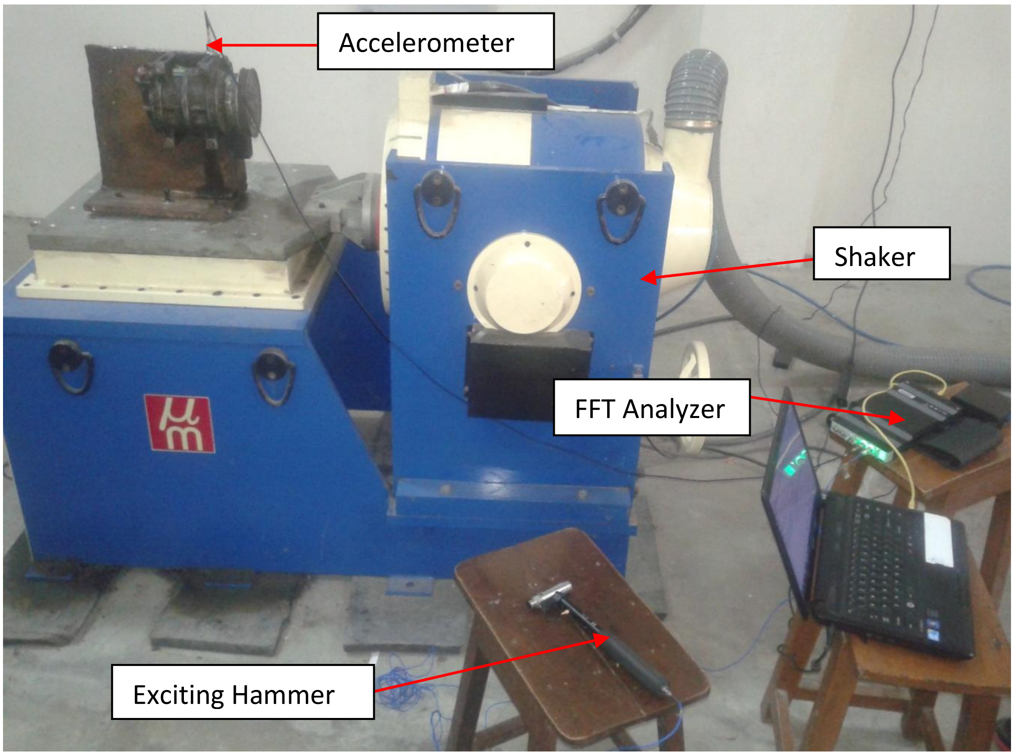


Figure 5.6: Experimental setup for modal testing

The main components of the setup are:

(i) **Accelerometer:** It is an electromechanical device that measures the proper acceleration given during vibration testing on shaker. It acts as a vibration sensor and give the results of the FFT analysis. In the present work, uniaxial directional accelerometer was used.

(ii) **FFT analyzer:** FFT is fast fourier transform which is used in measuring vibration of a component. It takes input from the accelerometer and converts them into the computer understandable form.

(iii) **Exciting hammer:** It is used to give excitation to the bracket mounted on the fixture. It is connected with the FFT analyzer.

(iv) **Pulse software:** Pulse software is used to visualize the results as per the requirement.

Firstly, the bracket was mounted on the fixture with the required boundary conditions. The hammer force was applied on the bracket to excited it. The limits of the force applied by the hammer were adjusted in the software for the reference purpose. The trigger of the hammer was tested as per the reference force in the software. Vibration occurs when the force applied by the hammer is more than the reference force. The natural frequency is the frequency corresponding to a peak, when the value of coherence is nearly 1. The peak can be seen from the amplitude vs. frequency graph. The modal testing was done on both original and optimized bracket.

5.1.4 Modal Testing on Original Bracket under Different Boundary Conditions

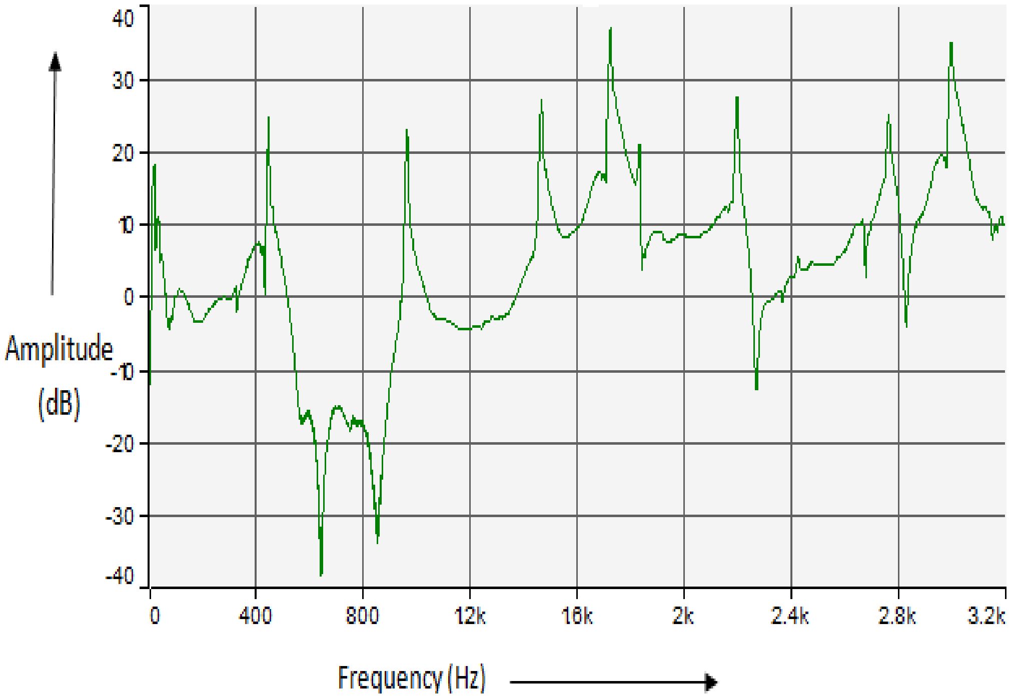
The modal testing was performed on original bracket under following boundary conditions.

(i) Free – Free condition

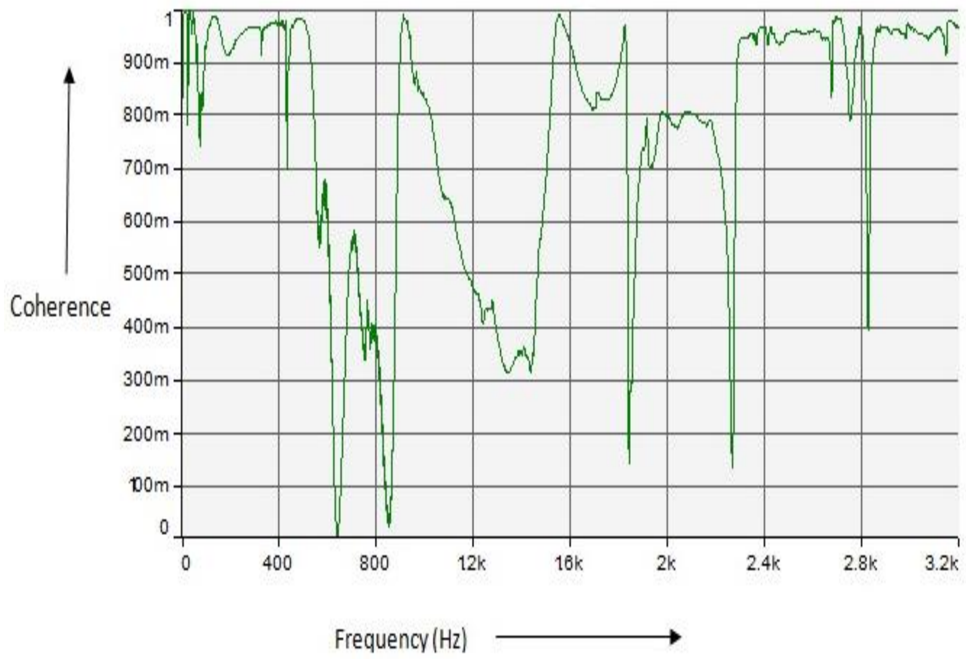
Under this boundary condition, the bracket was free to move in any direction. There was no constrained impose on this bracket. To minimize the damping from the ground, foam was used for this testing. Figure 5.7 shows the setup for free-free boundary condition.

Figure 5.7: Experimental setup during free-free condition of original bracket

The results for frequency response and coherence are shown in Figure 5.8.



(a)



(b)

Figure 5.8: Graphs plot during modal testing of original bracket under free-free condition: (a) Amplitude vs. Frequency; (b) Coherence vs. Frequency

The coherence function is used in conjunction with frequency response function. It indicates the quality of measurement of frequency response function and indicates the correlation of the response energy to the stimulus energy. In the presence of another signal *i.e.*, excessive noise or any other signal the quality of the network response measurement is poor. A zero value of coherence for any frequency indicates that there is no correlation between the stimulus signal and response. A value of 1 for a given frequency indicates that there is no interference at the given frequency *i.e.*, the response energy is 100% due to stimulus signal. So, to find out the natural frequencies of a bracket at a given condition, both the graphs are used. The peak points corresponding in amplitude vs. frequency plot are the natural frequencies provided these points should have coherence near to 1.

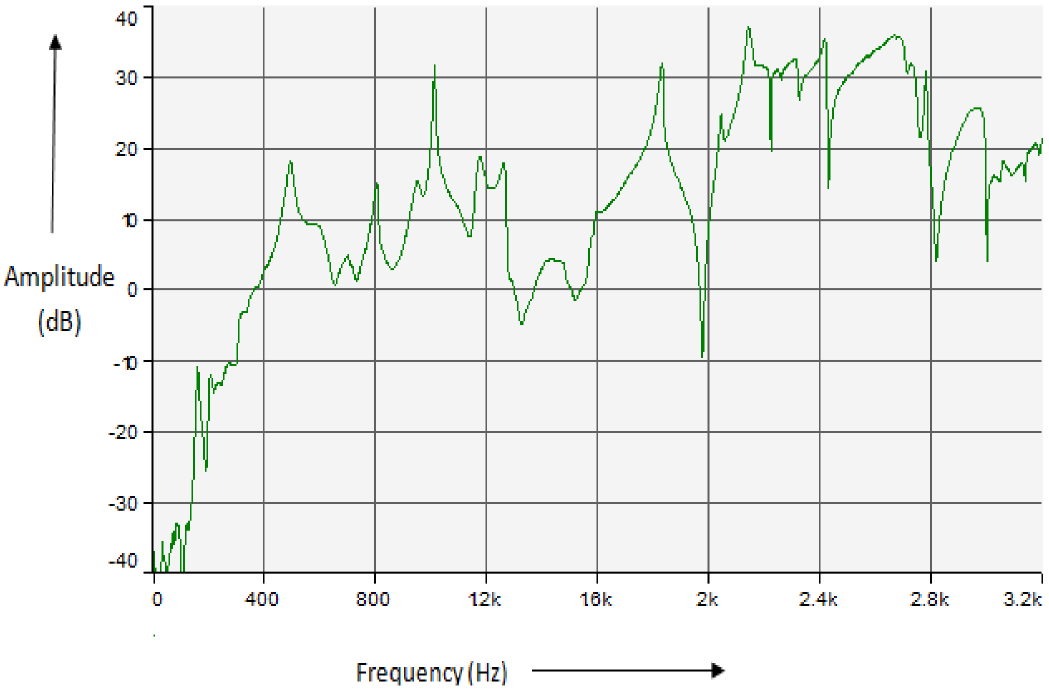
It can be seen from Figure 5.8 that the natural frequencies of original bracket at free –free condition are 442 Hz, 958 Hz, 1466 Hz, and 1720 Hz.

(ii) Original Bracket without Compressor

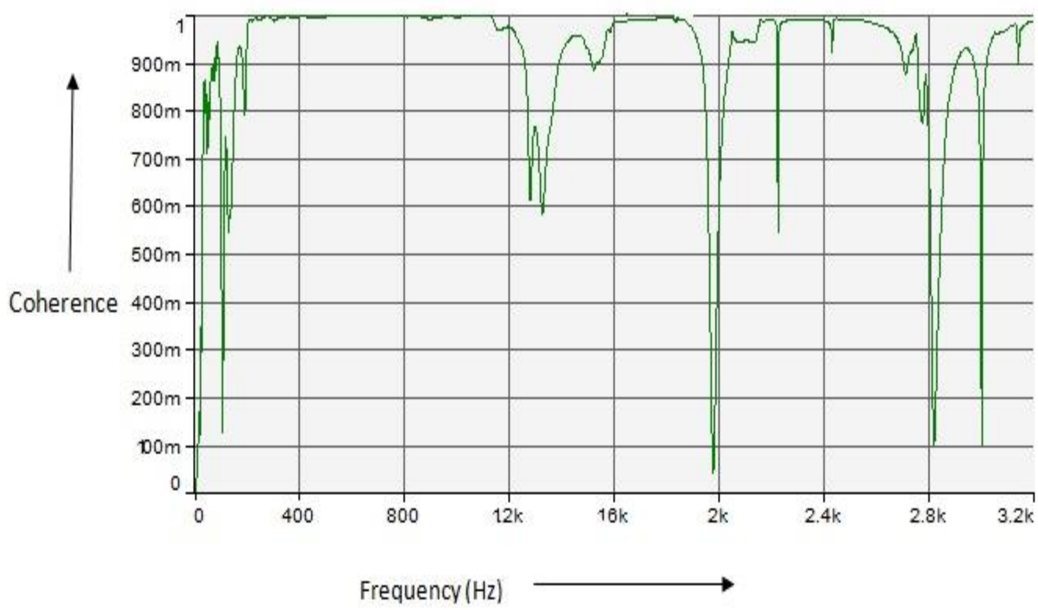
Under this boundary condition, the bracket without compressor was mounted on the fixture. The experimental setup for this condition is shown in Figure 5.9.

Figure 5.9: Experimental setup for original bracket without compressor

The frequency response and coherence graphs are shown in Figure 5.10.



(a)



(b)

Figure 5.10: Graphs plot during modal testing of original bracket without compressor: (a) Amplitude vs. Frequency; (b) Coherence vs. Frequency

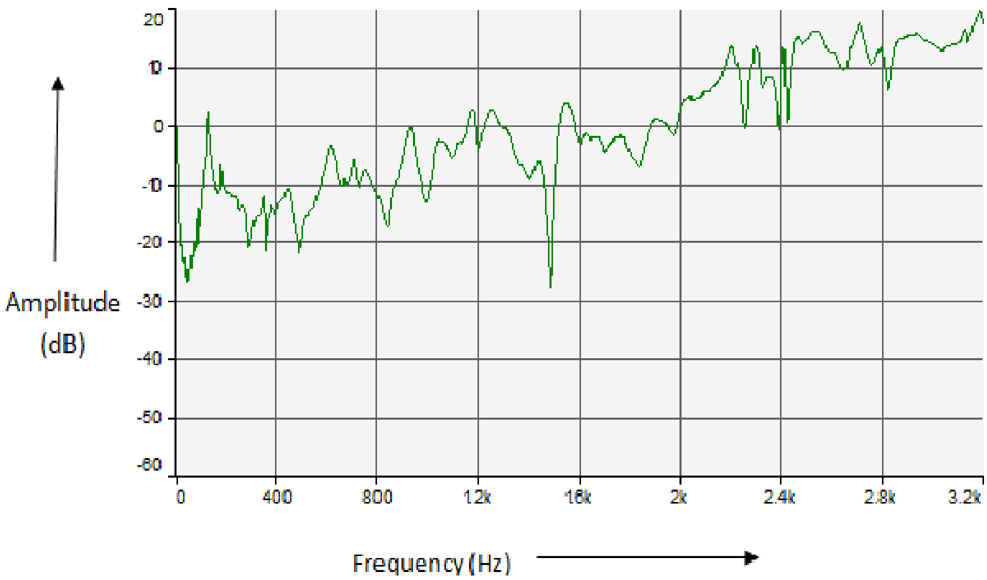
It can be seen from Figure 5.10, the natural frequencies of original bracket without compressor are 158 Hz, 492 Hz, 698 Hz, and 800 Hz.

(iii) Original Bracket with Compressor

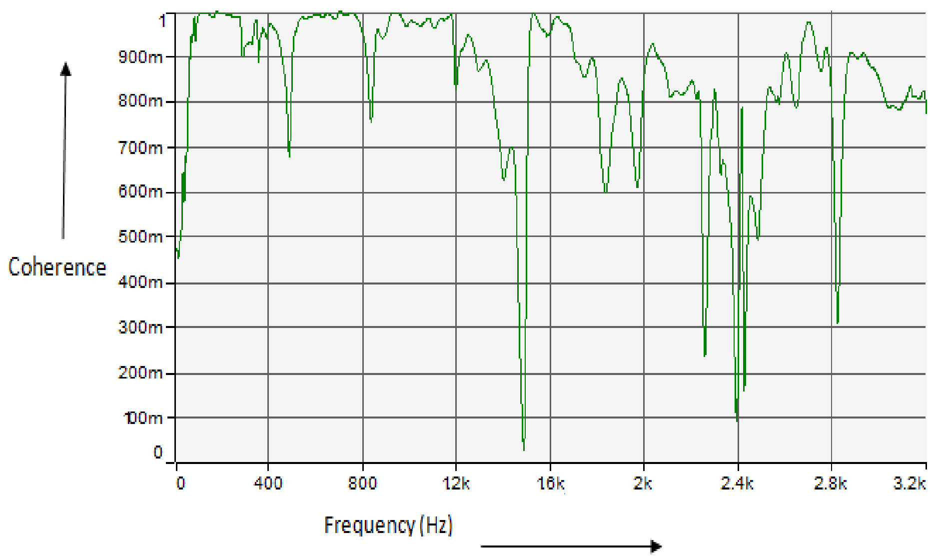
In this condition, the compressor was mounted on the bracket. This type of condition is the proper constraint condition. The experimental setup for this condition is shown in Figure 5.11.

Figure 5.11: Experimental setup for original bracket with compressor

The frequency response and coherence results are shown in Figure 5.12.



(a)



(b)

Figure 5.12: Graphs plot during modal testing of original bracket with compressor: (a) Amplitude vs. Frequency; (b) Coherence vs. Frequency

It can be seen from Figure 5.12 that the natural frequencies of original bracket with compressor are 80 Hz, 120 Hz, 172 Hz, and 338 Hz.

5.1.5 Modal Testing of Optimized Bracket under Different Boundary Conditions

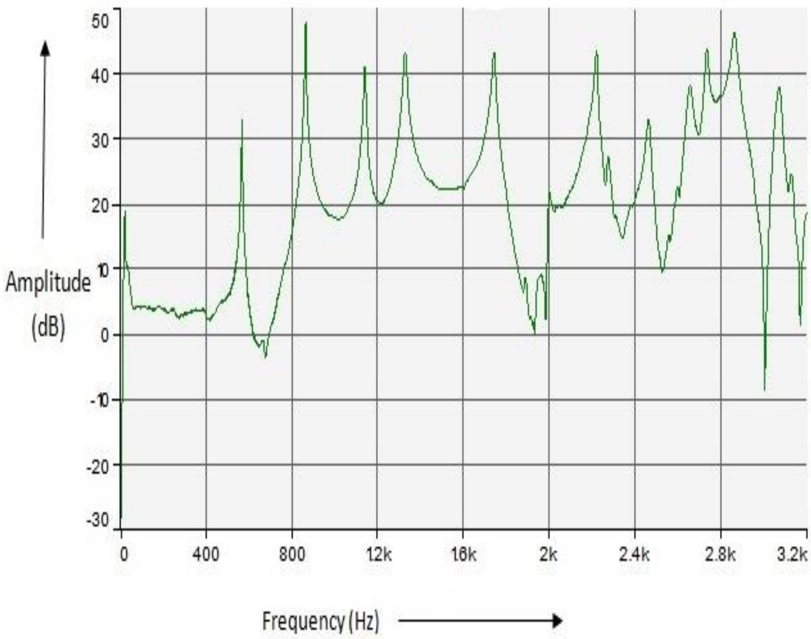
The same conditions were taken for optimized bracket to compare the results.

(i) Free – Free Condition

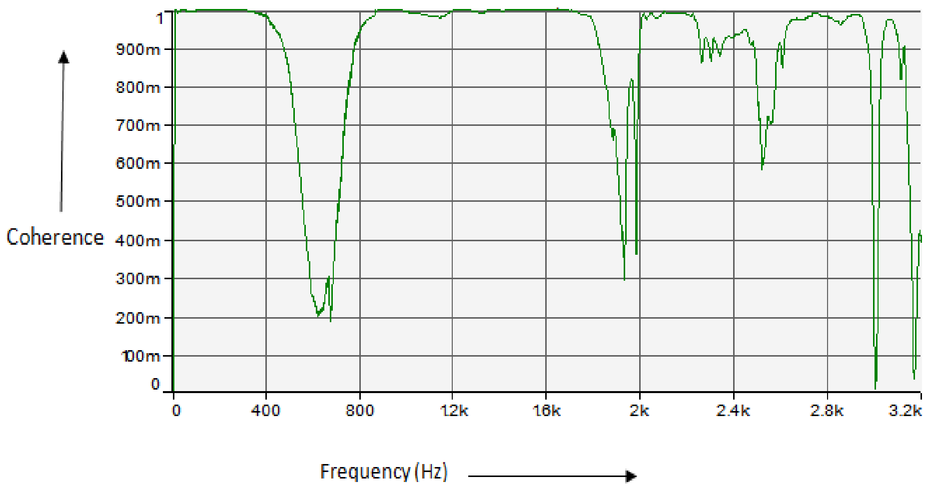
The experimental setup for optimized bracket is shown in Figure 5.13.

Figure 5.13: Experimental setup for optimized bracket under free-free condition

The frequency response and coherence results are shown in Figure 5.14.



(a)



(b)

Figure 5.14: Graphs plot during modal testing of optimized bracket under free- free condition: (a) Amplitude vs. Frequency; (b) Coherence vs. Frequency

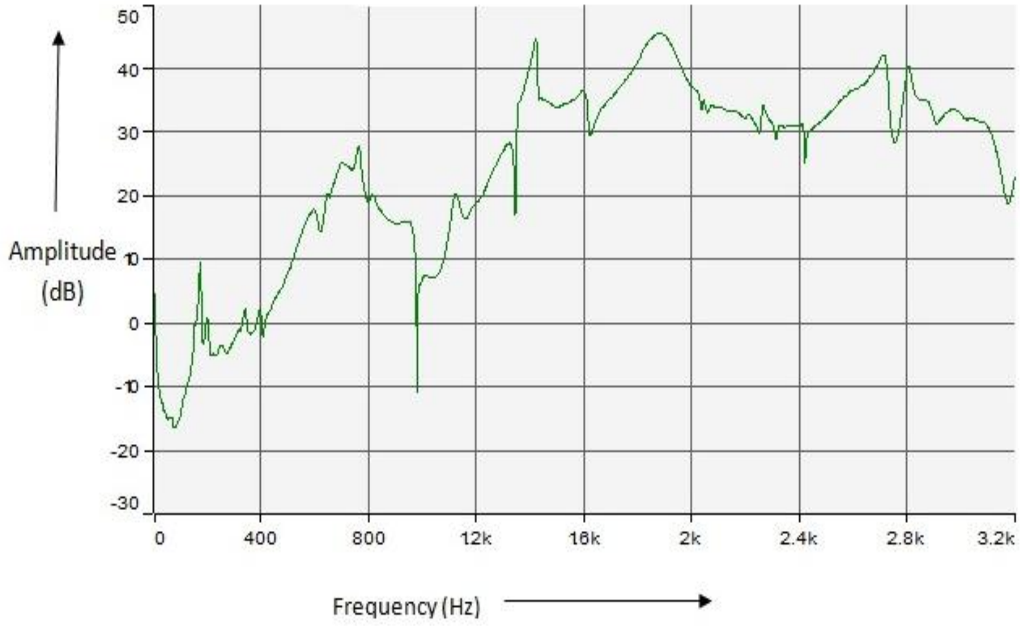
It can be seen from Figure 5.14 that the natural frequencies of optimized bracket under free- free condition are 562 Hz, 860 Hz, 1136 Hz, and 1326 Hz.

(ii) Optimized Bracket without Compressor

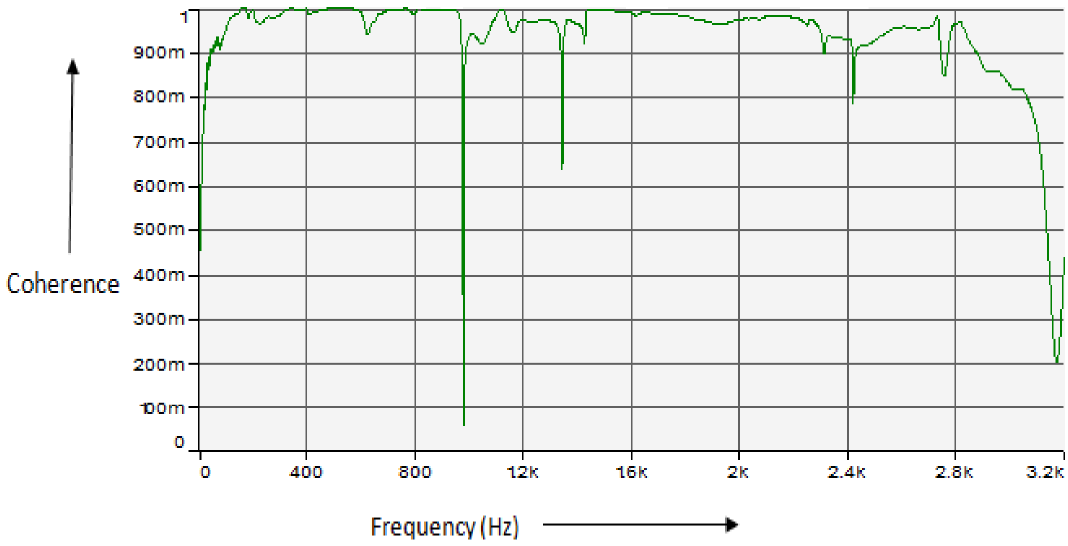
The experimental setup for the bracket is shown in the Figure 5.15.

Figure 5.15: Experimental setup for optimized bracket without compressor

Figure 5.16 shows the frequency response and coherence results of the bracket without compressor mounted on it.



(a)



(b)

Figure 5.16: Graphs plot during modal testing of optimized bracket without compressor: (a) Amplitude vs. Frequency; (b) Coherence vs. Frequency

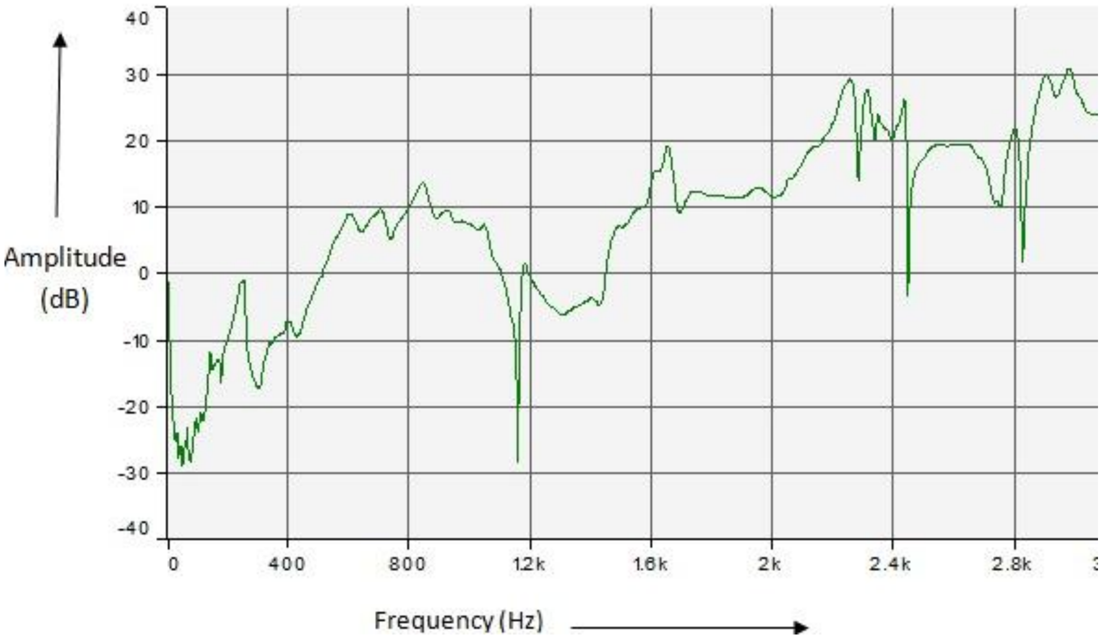
It can be seen from Figure 5.16 that the natural frequencies of optimized bracket without compressor are 172 Hz, 248 Hz, 360 Hz, and 590 Hz.

(iii) Optimized bracket with Compressor

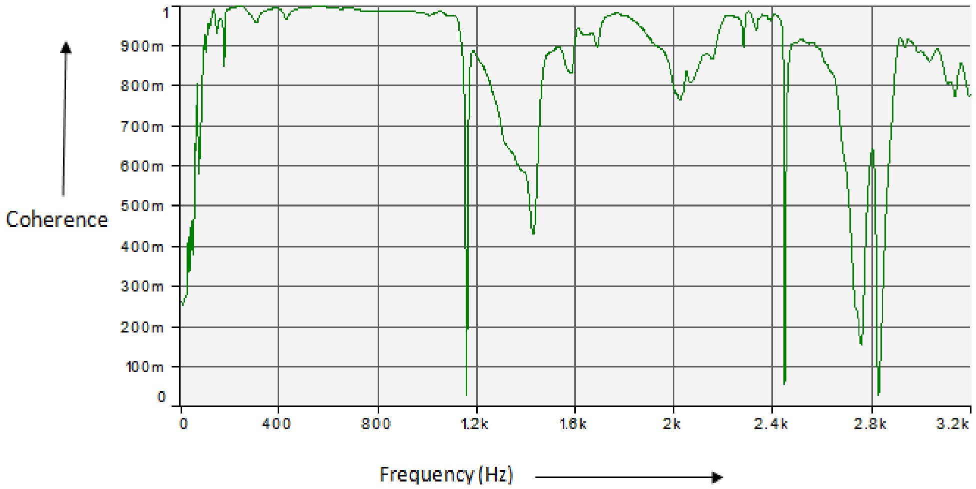
Figure 5.17 shows the setup for the testing of the bracket with compressor mounted on it.

Figure 5.17: Experimental setup for optimized bracket with compressor

Figure 5.18 shows the frequency response and coherence results for the bracket with compressor mounted on it.



(a)



(b)

Figure 5.18: Graphs plot during modal testing of optimized bracket with compressor: (a) Amplitude vs. Frequency; (b) Coherence vs. Frequency

It can be seen from Figure 5.18 that the natural frequencies of optimized bracket with compressor are 170 Hz, 210 Hz, 250 Hz, and 394 Hz.

5.2 Comparison of Results

The results of both original and optimized bracket were obtained under different boundary conditions. For the comparison of results, the condition with the compressor mounted on the bracket was taken. Table 5.2 shows the comparison of CAE and experimental results for the original bracket with compressor.

Table 5.2: Natural frequencies comparison of original bracket with compressor

Natural frequency	Using Simulation software	Experimental data
1^{st}	99.746 Hz	80 Hz
2^{nd}	116.21 Hz	120 Hz
3^{rd}	186.36 Hz	172 Hz
4^{th}	246.82 Hz	338 Hz

Table 5.3 shows the comparison of results for optimized bracket with compressor mounted on it.

Table 5.3: Natural frequencies comparison of optimized bracket with compressor

Natural frequency	Using Simulation software	Experimental data
1^{st}	168.69 Hz	170 Hz
2^{nd}	235.99 Hz	210 Hz
3^{rd}	265.67 Hz	250 Hz
4^{th}	343.49 Hz	394 Hz

It can be seen from Table 5.2 and 5.3 that there is fairly good correlation between the software and experimental results. Some variation in the results is due to mesh size, number of elements and complex geometry of the bracket. The environmental condition also affects the results that are obtained experimentally.

Table 5.4 shows the comparison of mass and the first natural frequency of the bracket at different optimization stages.

Table 5.4: Mass and First natural frequency of bracket at different stages

Parameter	Original	Shape optimization of original bracket	Base design for topology optimization	After Topology optimization of original bracket	After shape optimization and topology optimization both
Mass	1.630 kg	1.627 kg	2.35 kg	1.600 kg	1.590 kg
First natural frequency (simulated data)	99.746 Hz	100.72 Hz	186.26 Hz	158.81 Hz	168.69 Hz

The different optimization methods were applied at different stages to improve the design of a bracket. It can be seen from Table 5.4 that the natural frequency and mass of the bracket has been improved after performing both shape and topology optimization. The mass of a bracket reduced by 40 gm and the natural frequency has been improved from 99.746 Hz to 168.69 Hz.

CHAPTER 6 CONCLUSION

6.1 Conclusion

Static structural and dynamic analyses were performed on bracket used for mounting air-conditioning compressor in an automobile. The original bracket was optimized using shape and topology optimization. The simulation results of both original and optimized bracket were validated using the experimental setup. The simulation and experimental results had a very good correlation between them. It was observed that the mass of bracket was reduced by 40 gm with the natural frequency increasing from 80 Hz to 170 Hz which indicates that the optimized bracket has higher stiffness than the original design.

6.2 Scope for Future Work

The present work can be extended in the following directions:

a) Endurance and fatigue testing can be performed on the bracket.
b) Random response analysis can be performed.
c) Transient response analysis can be evaluated for the bracket.
d) Simulation analysis can be adopted for other mounting parts in an automobile.

PUBLICATION

[1] Sachin Kalsi, Daljeet Singh, J.S.Saini, "Optimization of Compressor Mounting Bracket of a Passenger Car", *International Journal of Automotive Technology*. (Communicated)

REFERENCES

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

[2] 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.

[3] Steffen Johnsen (2013), " Structural Topology Optimization : Basic Theory, Methods and Applications", *Thesis of Master of Science in Mechanical Engineering*, Department of Engineering Design and Materials, Norwegian University of Science and Technology.

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

[5] 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.

[6] 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.

[7] 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.

[8] Murali M.R.Krishna (2002), “Finite Element Topography and Shape Optimization of a Jounce Bumper Bracket”, *SAE Technical Paper*, Doi: 10.4271/2002-01-1468.

[9] 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.

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

[11] Eric A.Nelson (2003), “Draw Directions Constraints in Topology Optimization- A Practical Example”, *SAE Technical Paper*, Doi: 10.4271/2003-01-1306.

[12] Doo-Ho Lee, Jeong-Woo Chang and Chan-Mook Kim (2003), “Optimal Shape Optimization of an Air-Conditioner Compressor-Mounting Bracket in Passenger Car”,*SAE Technical Paper*, Doi: 10.4271/2003-01-1306.

[13] Chang-Seong Ko, Doong-Ho Yoo and Kyung-Whan Park (2003), “Design of Steering Column Bracket for Vibration”, *SAE Technical Paper*, Doi: 10.4271/2003-01-2747.

[14] Eduardo Castelo Branco Porto, Geraldo Minoru Kato, Rogerio Ribeiro and Walter Wilhelm Lorenz (2004), “Structural Optimization of a Rear Cabin Suspension Bracket and of Frame Air Filter Bracket”, *SAE Technical Paper*, Doi: 10.4271/2003-01-2747.

[15] 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.

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

[17] Vinicius Leal,Valdir Mendes Cardoso, Luciano Zambelli, Guilherme Cameiro and Rudinixon Bitencourt (2006), “Shape and Topology Optimization of Engine Mounts”, *SAE Technical Paper,* Doi: 10.4271/2006-01-2665.

[18] Xiao-Tong Pan, Doni Zonni, Guo- Zhong Chai, Yan- Qing Zhao and Cui-Cui Jiang (2007), “Structural Optimization for Engine Mount Bracket”, *SAE Technical Paper*, Doi: 10.4271/2006-01-2665.

[19] 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.

[20] Jeong Woo Chang and Young Shin Lee (2008), “Topology Optimization of Compressor Bracket”, *Journal of Mechanical Science and Technology*, vol. 22, pp. 1668–1676.

[21] S.K. Loh, W.M. Chin and Waleed F. Faris (2009), “Fatigue Analysis of Package Terminal Air Conditioner Motor Bracket under Dynamic Loading”, *Materials and Design*, vol. 30, pp. 3206–3216.

[22] 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.

[23] Gang-Won Jang,Min-Su Yoon and Jae Ha Park (2010), “Lightweight Flatbed Trailer Design by using Topology and Thickness Optimization”, *Sruct Multidisc Optim*, vol. 41, pp. 295–307.

[24] M.Ccagri Cevik, Emre Kanpolat and Martin Rebbert (2011), “Shape Optimization of Single Cylinder Engine Crankshaft”, *SAE Technical Paper*, Doi: 10.4271/2011-01-1077.

[25] 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.

[26] Marco Cavazzuti,Andrea Baldini,Enrico Bertocchi,Dario Costi,Enrico Torricelli and Patrizio Moruzz (2011), "High Performance Automotive Chassis Design a topology optimization based approach", *Struct Multidisc Optim*, vol. 44,pp. 45–56.

[27] 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.

[28] 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.

[29] Matteo Bruggi and Pierre Duysinx (2012), "Topology Optimization for Minimum Weight with Compliance and stress constraints", *Struct Multidisc Optim*, vol. 46,pp. 369–384.

[30] 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.

[31] C.H. Park, H.J. Shim, D.H. Choi, J.K. Kim and S.M. Lee (2012), "Shape Optimization of Rubber Isolators in Automotive Cooling Modules for the Maximization of Vibration Isolation and Fatigue Life", *International Journal of Automotive Technology*, vol. 13, pp. 61–75.

[32] Parag Memichand Jain and Satish Pavuluri (2013), "Experimental and Finite Element Analysis of Bogie Suspension Brackets", *SAE Technical Paper*, doi: 10.4271/2013-01-2789.

[33] Eduardo Domingo Morales, Sergio Ferreira Alexandre and Leandro Vieira da Silva Macedo (2013), “Optimization Methods Applied to Development of Vehicle Structures”, *SAE Technical Paper*, Doi: 10.4271/2013-36-0455

[34] 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.

[35] T. D. Tsai and C. C. Cheng (2013), “Structural Design for Desired Eigenfrequencies and Mode shapes using topology optimization”, *Struct Multidisc Optim*, vol. 47, pp. 673–686.

[36] Mulla Salim Ha, Yadv Sanjay Db, Dhananjay Shindec and Gaurav Deshpanded (2013), “Importance of Federal Motor Vehicle Safety Standards 207/210 in Occupant Safety - A Case Study”, International Conference On Design And Manufacturing, IConDM 2013,*Procedia Engineering*, vol.64, pp.1099 – 1108.

[37] 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

[38] K A Zakria, S Abdullah, M J Ghazali, M Z Nuawi, S M Beden and Z M Nopiah (2014), “Fatigue Strain Signal Behaviour and Damage Assessment of an Engine Mounting”, *Journal of Scientific and Industrial research*, vol. 73, pp.112–116.

[39] 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

[40] http://www.99rpm.com/tata/ac-compressor-3-0l-dicor safari.html

[41] Srinath Viswanathan, Diran Apelian, Raymond J. Donahue, Babu DasGupta, Michael Gywn, John L.Jorstad, Raymond W. Monroe, Mahi Sahoo, Thomas E. Prucha, Daniel Twarog, Steve Lampman, Charles Moosbrugger, Eileen DeGuire, Madrid Tramble, Ann Britton, Diane Whitelaw, Kathryn Muldoon, Scott D. Henry, Bonnie R. Sanders, Elizabeth Marquard, Heather Lampman and Buz Riley (2008),"ASM Handbook Volume 15 Casting", *ASM International*,US.

[42] John Bird (2012), "Science for Engineering", *Routiledge*, US.

[43] D. S. Ramakrishna, K. Sabeel Ahmed and C. D. Ashok Kumar (2010), "Proceedings of National Conference on Advances in Mechanical Engineering", *Allied Publishers Private Limited*, India.

[44] Yasuo O., Nagashima H. and Tanahashi (1995), "Vehicle technology handbook tests and evaluation", *Society of Automotive engineers of Japan*.

[45] http://altairenlighten.com/2011/12/sizing