DYNAMIC ANALYSIS AND OPTIMIZATION OF A CHASSIS MOUNTED BRACKET OF A TRUCK

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

MASTER OF ENGINEERING
IN
CAD/CAM ENGINEERING

Submitted By

Bikramjeet Singh

Roll No. 801481007

Under the Guidance of

Mr.Daljeet Singh
Assistant Professor,
Mechanical Engineering Department
Thapar University ,Patiala

Mr.Ashish Purohit
Assistant Professor
Mechanical Engineering Department
Thapar University ,Patiala

DEPARTMENT OF MECHANICAL ENGINEERING

Thapar University

Patiala-147004, Punjab

July, 2016

DECLARATION

I hereby declare that the work done in this Thesis entitled, "**Dynamic Analysis and optimization of a Chassis Mounted Bracket of a truck**" is an authentic record of work carried out by me for the award of degree of **Master of Engineering** in **CAD/CAM Engineering** in Mechanical Engineering Department of **Thapar University, Patiala**, under the supervision and guidance of **Mr.Daljeet Singh,** Assistant Professor of Mechanical Engineering Department, Thapar University ,Patiala **and Mr.Ashish Purohit**, Assistant Professor of Mechanical Engineering Department, Thapar University, Patiala.

No part of the matter embodied in this report has been submitted in part or full to any other university or institute for the award of any degree.

DATE: 12/07/2016

Bikramjeet

Bikramjeet Singh

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

Daljeet Singh

Mr.Daljeet Singh
Assistant Professor, MED
Thapar University, Patiala

Mr.Ashish Purohit
Assistant Professor, MED
Thapar University, Patiala

Countersigned by:

(Dr. S. K. Mohapatra)
Head, Mechanical Engineering Department
Thapar University, Patiala

(Dr. S. S. Bhatia)
Dean of Academic Affairs
Thapar University, Patiala

Dedicated to my

Father and Mother

Acknowledgements

I would first like to thank my thesis advisors **Mr.Daljeet Singh**, Assistant Professor of Mechanical Engineering department and **Mr.Ashish Purohit** Assistant Professor, of Mechanical Engineering department, Thapar University, Patiala for their valuable guidance .They regularly help me whenever I had some doubt about my research or writing.

I am also thankful to **Dr. S.K Mohapatra,** Professor and Head, Department of Mechanical Engineering for providing the facilities and support for finishing the thesis work.

I am grateful to my parents for providing me with unfailing support and continuous encouragement throughout my years of study and through the process of researching and writing this thesis. This accomplishment would not have been possible without them. Thank you.

I would also like to thanks Dr. S.P Singh, Professor, Mechanical engineering department, IIT Delhi, Dr. Ashish K. Darpe, Associate Professor, Mechanical engineering department, IIT Delhi, and Mr. K. N Madhu, Lab technician, Vibration Research Lab, IIT Delhi for their support to provide the equipment required to perform the experiment in this thesis.

Bikramjeet Singh

Registration No. : 801481007

Abstract

The advancement of Computer Aided Engineering (CAE) makes industry to perform dynamic/static analysis of vehicle components in actual working environment. In the present work, a chassis mounted bracket of Eicher truck has been considered for dynamic analysis using the actual road vibration excitation. The bracket is used as a foundation to support the heavy spare tyre and mounted on the chassis. The dynamic forces subjected on the bracket can generate high stresses in the bracket, which also influences by the speed and different road conditions. High stresses may lead to structure failure or fatigue failure of the bracket. Generally, the components are overdesign to avoid such failures due to dynamic loading, which in turn increases the overall weight of the component/vehicle and affect the performance of the vehicle. In this study, a methodology is developed to dynamically analyse the mounting bracket using real time vibration excitation induced on the support structure of bracket. The end effect of the excitation on the bracket is further used to optimized the bracket.

The mounting bracket has been designed and analysed using commercial packages as CREO and ANSYS Workbench respectively. Real time vibration signal is acquired in the form of acceleration on the base of bracket and used as an input dynamic force to the support of the bracket to perform transient dynamic analysis. The resultant stresses due to static structural and dynamic analysis are compared with yield strength of material to check the design feasibility. From the different analysis, it is noted that the chassis mounted bracket is overdesigned and, hence, optimization of bracket is performed to reduce its mass, while keeping the strength unaltered. Other FEA analysis is also performed on bracket like static structural analysis to check the total deformation in bracket due to dead weight of tyre and modal analysis to find the natural frequency of bracket. Using current methodology, a 12.25 % reduction in the bracket has been achieved.

Key Words: Finite Element Analysis; Transient analysis; Dynamic analysis; Road vibration; Chassis mounted bracket; Power spectral density; Optimization.

Table of Contents

List of Figures

List of Tables

Acronyms

FEA	Finite Element Analysis
PSD	Power Spectral Density
RMS	Root Mean Square
CG	Center of Gravity
NI	National Instrument
S_{YT}	Yield Strength in Tensile (MPa)
S_{UT}	Ultimate Strength in Tensile (MPa)
DOF	Degree of Freedom
CAD	Computer Aided Design
DAQ	Data Acquisition

Chapter 1

INTRODUCTION

An automotive heavy vehicle like truck requires a number of structural parts to hold various components. Engine mounting brackets, suspension bracket, chassis mounted brackets etc. are some of the example of such structural parts. These structural brackets are subjected to various kind of excitation originated from various sources such as periodic excitation from the engine vibration, unbalance in the tyre assembly, random loads generated from the road profile etc. Designing of these structural components against the dynamic loading is an important aspect in overall design of the vehicle. Dynamic loading forces structural member to vibrate, which may lead to fatigue failure of the component. It also influences the performance of the vehicle. In many cases, the components are overdesign to avoid failure due to dynamic loading, which in turn increases overall weight of the vehicle and affects the performance of the vehicle. Selection of a heavy bracket also increases the cost. A significant research is underway to optimize the weight of such structural component. The current investigation is also related to the optimization of an automotive bracket against the dynamic loading measured from the actual road.

1.1 Vibration in mechanical system

Vibration is an oscillating motion of a system or a body or an assembly of bodies about its mean or equilibrium position. Vibration occurs in the structure or any system when it is displayed from its equilibrium position by an external excitation. A typical vibrating system is consist of an inertial mass and an element that provide restoring force. In most of the application, vibrations are an undesirable, it may influence in many ways as:

- Increases Stress level
- Decreases Fatigue life
- It leads to the discomfort to passengers in vehicles

One of the main causes of failure in the mechanical components of vehicles is vibration. Therefore, vibration is one of intense area of modern engineering research. Vibration characteristics of physical model are the main design parameters in product design cycle.

Conversely, in some of the front, vibration is useful for machines or structures like Rock driller which uses vibration energy to drill the components, Electro dynamic shaker for creating the vibration testing environment, vibration exercise machine to relax the muscles, vibratory tumbler for cleaning rusty metal parts etc.

An actual vibratory system is complicated and difficult to analyse. Therefore, in the mathematical modelling of the system, a number of assumptions are considered for simplification. Many vibration systems consist of single component or multiple components. Dynamic characteristics of component are defined by characteristics like damping ratio, natural frequency etc. The single component dynamic vibration characteristics can be found by experimental, numerical and analytical method. After defining the component characteristics, actual system can be represented by a mathematical model. Mathematical models can be divided into two types:-

1) Discrete or lumped system which is represented by ordinary differential equations.
2) Distributed or continuous system which is represented by partial differential equations

1.1.1 Methods to analyse any physical vibration system

There are three methods for analysing any physical vibration system:-

a) Analytical method
b) Numerical method
c) Experimental method

a) Analytical Method

This method is based on classical approach. Physical system is represented by mathematical equations and these equations are solved to find the value of system parameter at any point on physical system. This method is very difficult as compared to other two method and required lengthy hand calculations. This method is used to solve system having less Degree of freedom, mostly one or two Degree of freedom (DOF) system. This method includes analysis of problems like stress and strains in cantilever beam, deflection in simply support beam, estimating of natural frequency of beam (Eigen value problem) etc.

b) Numerical Method

It is used to calculate the approximate solution of the problem. It is also based on mathematical equations but these equations are solved for limited points to find the value of system parameter. For other points solution is approximated by Shape functions. In this method continuous domain is converted into discrete domain. This is very robust method and can be used for analysing a complicated problem. It is based on the discretization approach, which may be Finite element method (FEM), Finite volume Method (FVM) etc. This method is basics of number of analysis software like ANSYS, Hyperworks etc. Time required to solve a complicated governing equations is less. This method can be used for solving multiple DOF problems.

c) Experimental Method

It is based on physical testing of actual prototype. It gives most accurate results and accuracy of the other methods can be checked as comparative to experimental method. It is quite expensive method and requires equipment to perform an experiment. Experimental study is the last investigation that is essential to give final confirmation to design. By carrying out this method, it is confirmed that the design is ready for manufacturing or not.

1.1.2 Types of vibration

Vibrations can be categorized as free and forced vibration. Figure 1.1 shows the classification of vibration in terms of a flow chart.

a) Free vibration

In the free vibration, the system exhibits oscillation in the absence of any excitation. In the free vibration, the system vibrates on its natural frequency.

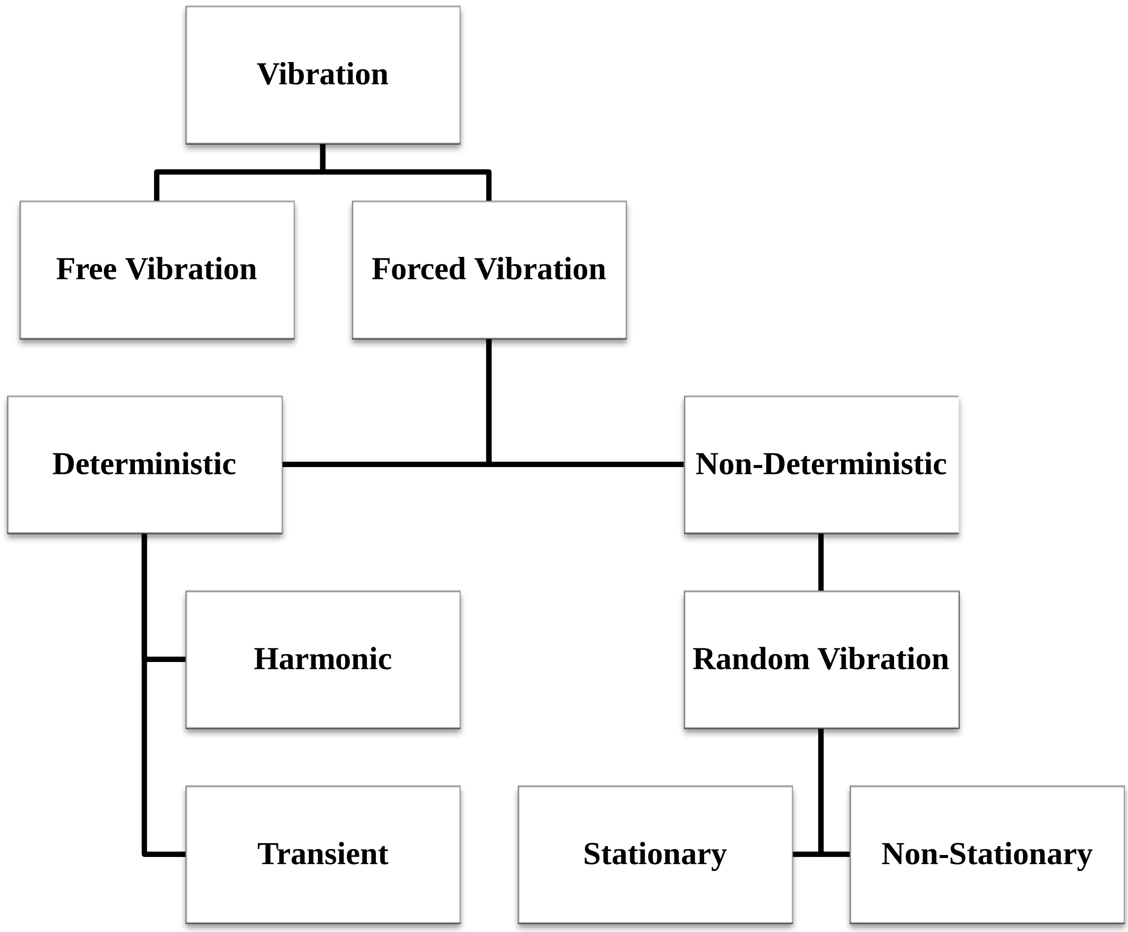


Figure 1.1: Different types of vibration

b) Forced Vibration

In the forced vibration state, the system is excited through an external force and the system vibrate at the frequency of the external excitation rather than in its natural frequency as observed in case of free vibration.. Example:-Vibration in engine mountings, Base excitation in vehicle due to road etc.

- **Deterministic Vibration**

If the external vibration amplitude and phase is precisely known at any particular instant of time, then the vibration is called deterministic vibration. Example: -

Vibration in component due to some harmonic excitation (sine or cosine function), Vibration in a structure attached to engine.

- **Non- Deterministic Vibration**

The vibration amplitude and phase is not precisely known at any particular instant of time. Excitation is random in nature. Statistical models are used to find the solution of this type of problem. Example: - vibration in chassis in a moving vehicle on road.

- **Random Vibration:** It is vibration in which amplitude of the vibration is non-deterministic. One cannot precisely predict the value of vibration at any instant of time. Like if vibration was measured on any component of vehicle moving on road, you cannot precisely predict the amplitude and phase of vibration at any particular instant of time. Response of a system to random excitation is also random. Figure 1.2 shows the random vibration data measured on truck chassis.

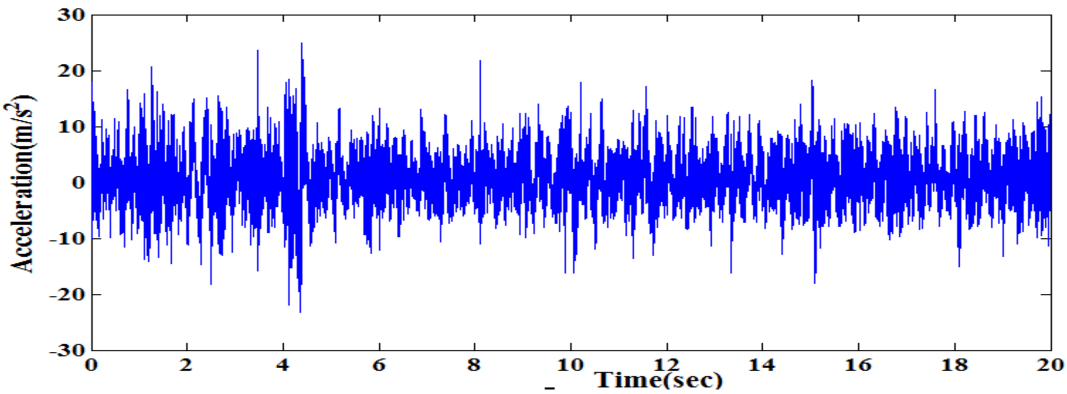


Figure 1.2: Random vibration data measured on truck chassis

Examples of Random vibration

- Motion of building during earthquake
- Vibration in body of rocket during lift off
- Vibration in ship due to wave motion

Characteristics of Random Vibration

- It is non-periodic in nature.
- It is composed of spectrum of frequencies.

- Cannot be defined by a deterministic mathematical function.
- No fixed ratio between its peak and RMS values

Fourier Transformation, Fast Fourier Transform (FFT) and Power Spectral Density (PSD):-

Fourier Transform is used to transform a time domain vibration signal into frequency domain. It characterizes a time signal into a set of sine functions and each set having a distinctive amplitude, frequency and phase. These days the more standard way to represent a random vibration signal in frequency domain as a Power Spectral Density (PSD) functions. Moreover, in many design standards the data about random vibrational signal is given in PSD. The main purpose of converting the signal into frequency domain is to get the information about behaviour of signal with respect to frequencies. Frequencies accountable for resonance of structure can be easily detected, using this info.

In this thesis, the vibration data was recorded by using data acquisition system. In data acquisition system a computer is used that records the data in discrete form. This discrete data is converted into frequency domain by using discrete form of Fourier transform. A fast algorithm on the basis of discrete Fourier transform was developed which is popularly known as 'Fast Fourier Transform' (FFT).

The FFT of a signal gives the output in the form of complex number pertaining to frequency which comprises the information about amplitude of sinusoidal wave and its initial phase angle. But on the other hand PSD keeps only amplitude value. The amplitude of sine wave as per frequency is more important in most of the engineering applications. Figure 1.3 shows a time domain signal and its corresponding PSD signal in frequency domain.

Random Vibration Types

a) **Stationary vibration**
   If the statistical properties of random vibration signal like standard deviation, mean etc. remain constant relating to time it is called stationary vibration. It is an ideal concept.

b) Non-Stationary Vibration

As contrary to stationary vibration if statistical properties change with time the signal is called non-stationary vibration signal.

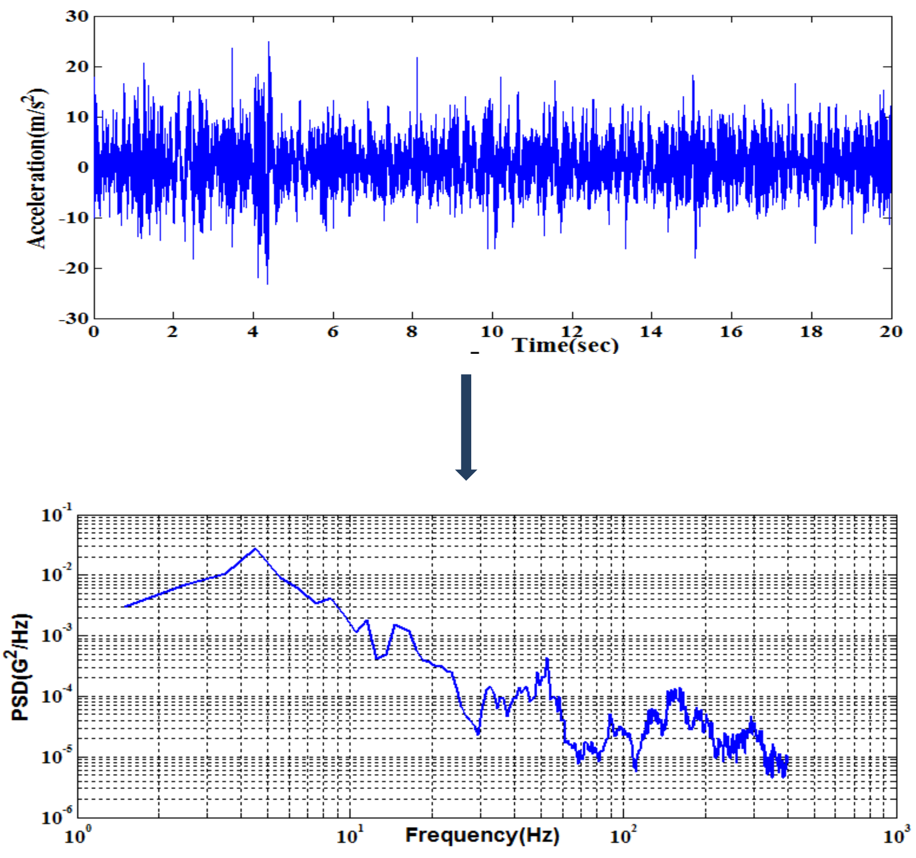


Figure 1.3: Conversion of Time domain signal into frequency domain

1.2 Different types of Finite Element Analysis (FEA)

The different types of FEA analysis used in this study are:

1.2.1 Linear static structural analysis

It is the first basic FEA analysis performed on structure to find its static behaviour. It is used to find the effect of steady state load on the body under consideration. Steady state means external force is varying very slowly with respect to time or there is no variation with respect to time i.e., dead weight of the tyre mounted to bracket in this study. In this analysis the damping and inertia effects are ignored. The equation of motion of a vibratory is written as

$$[M]\{\ddot{Y}\} + [C]\{\dot{Y}\} + [K]\{Y\} = \{F(t)\} \tag{1.1}$$

Where [M] = Structural mass matrix

$\{\ddot{Y}\}$ = Nodal acceleration vector

$[C]$ = Damping matrix

$\{\dot{Y}\}$ = Nodal velocity vector

[K] = Structural stiffness matrix

$\{Y\}$ = Nodal force vector

$\{F(t)\}$ = External applied force vector

In the above equation term $[M]\{\ddot{Y}\}$ represents force due to inertia, $[C]\{\dot{Y}\}$ represents damping force, $[K]\{Y\}$ represents the force due to spring stiffness force and $\{F(t)\}$ represents the external force.

In linear static structural analysis, damping and inertia forces are neglected, so the equation (1.1) is simplified as

$$[K]\{Y\} = \{F(t)\} \tag{1.2}$$

Assumptions considered in equation (1.2) are as follows:

a) [K] is constant due to linear elastic behaviour of material
b) Small deflection theory is used
c) Force is varying very slowly with time or force is constant

Equation (1.2) is used in static structural analysis to find the stresses, strains, displacement etc.

1.2.2 Modal analysis

This analysis is the first step to know about the basic dynamics of the system. It is used to find the most important characteristic of vibration as natural frequency and corresponding mode shape. These basic parameters are dependent on boundary condition, structural properties and material properties of component. The number of natural frequency of the component is directly proportional to the degree of freedom of the component. Any kind of damping for component is neglected during modal analysis. Moreover, an external force is not considered on component for modal analysis. So the equation of motion (1.1) is reduced to

$$[M]\{\ddot{Y}\} + [K]\{Y\} = 0 \tag{1.3}$$

This equation is also called equation of motion for free undamped vibration. To solve this equation harmonic solution is assumed as given in equation (1.4)

$$\{Y\} = \{\varphi\} \sin \omega t \tag{1.4}$$

Where $\{\varphi\}$ = Eigen vector or mode shape

ω = represents the circular natural frequency

By differentiating the equation (1.4) and substituting in the equation (1.3) gives the following results

$$-\omega^2[M]\{\varphi\} \sin \omega t + [K]\{\varphi\} \sin \omega t = 0 \tag{1.5}$$

Above equation after simplification becomes

$$([K] - \omega^2[M])\{\varphi\} = 0 \tag{1.6}$$

Equation (1.6) is called eigenvalue equation.

$$(A - \lambda I)x = 0 \tag{1.7}$$

Equation (1.6) is compared with basic eigenvalue equation as shown in the equation (1.7)

So, $\lambda = \omega^2$ and it is called eigen value and $\{\varphi\}$ is called eigen vector.

$$f_i = \frac{\omega_i}{2\pi} \tag{1.8}$$

The natural frequency of a structure is given by equation (1.8) .Where f_i represents the i^{th} natural frequency and $\omega_i = \sqrt{\lambda}$

1.2.3 Transient analysis

Transient analysis is done to find the dynamic solution of system due to time varying loads. These loads can be in the form of acceleration, velocity, displacement etc. The damping and inertial forces are considered in this analysis. If the system is considered to be linear, then the output of the transient analysis i.e. stresses strains etc. in the same form of the input mean varying with respect to time. Basically two methods are used to calculate the response in transient analysis .These two methods are:

a) Full method

It solves the full system equation without any approximation to find the solution of the problem that's why it takes very long time to find the solution. All types of non-linearity's can solve by this method. Mode analysis result not required prior to these analysis. This method is recommended for finding the accurate solution.

b) Modal Superposition method

It is approximate method used to find the approximate solution. It finds the response by summing the correct fraction of mode shape of lower frequencies. This method takes comparatively very less time to find the solution. It is best suitable for linear system and not accepting the non-zero displacement boundary condition.

Comparison between full method and modal superposition method is given in Table 1.1.

Table 1.1: Comparison between Full and Modal Superposition method

Sr. No.	Full Method	Modal Superposition Method
1	Accurate result	Approximate result
2	It take very large time to find the solution than MSUP	It take very less time than full method
3	All non-linearities are allowed	Generally use for linear model
4	Support all types of load and boundary condition	Support all type of load except non-zero displacement boundary
5	It solve full system equation simultaneously to find the solution	It solve uncoupled system of equation by doing a linear combination of lower mode shape

6	It is used for small part(less DOF)	It could be used for medium and large parts
7	It not require prior modal analysis	It require prior modal analysis

1.3 Thesis Outline

The main body of thesis comprises of seven chapters. The following points concisely explain about the thesis chapters.

Chapter 2 consists of literature survey about static and dynamic analysis of various mechanical structures.

Chapter 3 discusses about gaps in literature, objectives of thesis and methodology to accomplish the objectives.

Chapter 4 presents a methodology to acquire the real time vibration signals from bracket during the motion of a truck. Then the graphs of vibration signal acquired in time domain and their corresponding graph in frequency domain are shown. At last shows the comparison between different vibration signals is presented.

Chapter 5 consisting of various steps to perform the static and dynamic analysis of chassis mounted bracket in ANSYS. Various steps relate to preparation of CAD model, Finite element model, mesh generation, an application of boundary conditions for bracket etc. are discussed. This chapter discusses the results of static and transient analysis for bracket.

Chapter 6 focuses on the optimization of bracket. This chapter also discusses about the results of static and dynamic analysis for optimized bracket.

Chapter 7 represents the conclusion of the thesis and suggests ways of further extension of research work.

Chapter 2

LITERATURE REVIEW

2.1 General

In the present chapter, various past investigations carried out on the static and dynamic analysis of various types of mounting bracket and mechanical structure are discussed.

2.2 Literature Summary

Jadhav *et al.* [1] performed a static structural analysis and modal analysis of engine mounted bracket. For this study two different materials (magnesium alloy and aluminium alloy) were preferred for FEA analysis .Analysis demonstrates that a bracket weight reduced immensely by using magnesium alloy as a material as compared to aluminium alloy and natural frequency of bracket is more as compared to aluminium alloy. Magnesium alloy has good cast ability, low density, better welding properties, higher mechanical strength and corrosion resistance. According to results it concluded that magnesium alloy is a better material for Engine mount.

Melnikovs *et al.* [2] performed the dynamic analysis and optimization of automotive Gage Panel. To reduce the computation time, only important parts of assembly of automotive gage panel were considered for FEA analysis and other non-important parts which not greatly contribute to the gage panel dynamics are neglected. Static analysis result shows that the maximum stress occurred in lower bracket and at position where screen component is fixed. Harmonic analysis result shows that the maximum response for GP assembly occurred at first and third frequency. Response for harmonic analysis was calculated for only 7 critical points on GP assembly to reduce the computational time. These 7 critical points are selected on the basis of static analysis result and prior experience. Transient and random vibration analysis also done on bracket .These all analysis result is used to do the meta model based optimization of bracket of GP panel. The better design of bracket with variable thickness was accomplished by this method.

Agostinacchio *et al*. [3] analytically calculated the dynamic load generated by interaction between road and wheel of three vehicle types (bus, truck, and car) by using quarter car model. Road PSD for input to Quarter car model is generated by using ISO 8608 standard in matlab software. They found that vehicle speed has less effect on generation of vibration; longitudinal irregularity of road was the biggest factor for vibration generation. Increase in vehicle speed increases the frequency content of the excitation at the base of vehicle. The frequency of vertical motion for vehicle body occurred in the range of 1.5 to 4 Hz and for wheels in the range of 10 Hz. Due to this greater difference in frequencies the vehicle body motion is not dependent of wheel motion. The dynamic load value generated by truck is maximum and by car in minimum. Value of vibration is corresponding to dynamic load value.

Wolf Steiner *et al*. [4]proposed a new PSD-Analysis method for calculating the load spectra for fatigue life for random vibration. This method is computationally very effective and load spectra results were correct as come from in time domain method. Fatigue load spectra calculation method in frequency domain is limited to Gaussian distributed signal and deviation from Gaussian distribution decrease the accuracy of the result. In new proposed method the non-Gaussian distributed signal is converted into small Gaussian distributed signals. After conversion, a load spectrum is calculated by using frequency domain method (e.g. Dirlik). An accuracy of the new method was checked by apply it on axle mounted and bogie mounted components of railway vehicles.

Xu *et al.*[5]demonstrated by Steinberg Three interval method and Miner Cumulative Theory whether a particular component endured for required life cycles. Actuating cylinder parts of aeroplane landing gear system were chosen as research problem. Modal analysis was done on piston part. Then flight load spectrum as psd was an input to random vibration module in ANSYS software. The output of random analysis was stresses. These stresses were divided into three intervals as said by Steinberg. These stresses further used by Miner cumulative theory for calculating the fatigue life and this method is easily to think, without extra material fatigue performance test and convenient for engineering use

Koushik [6] analysed the engine mounting bracket of 2 tonne mini excavator of Tata Hitachi. Stresses were calculated in the bracket which was due to static load, self-weight and vibration load coming on the bracket. Vibration load in the form of multi axial accelerations on bracket was calculated by using data loggers. These acceleration data were used as input to do the transient dynamic analysis. This Analysis result shows the same critical failure areas as shown by simulation in software and field report. By analysing the filed failure data and hyperworks software data, design modifications was done on the bracket by adding additional ribs on the top of bracket to increase the stiffness of the bracket which increases its service life.

Mucchi *et al.*[7] carried out the Experimental Modal Program on different types of brackets under difference constraint conditions to make a damping database. The brackets with the same constraint condition show similar values of modal damping irrespective of shape. These damping values can be very important for the virtual prototyping phase to carry out dynamic analysis of similar brackets for which no test data was available.

Kagnici *et al.*[8] developed a new fatigue method which can predict the damage in automotive vehicle in less time. A mini bus was considered for this study .FEA model of this bus was created in Hyper mesh software which include all the critical parts like vehicle frame, front and rear suspension etc. and non-critical parts are represent as point mass like baggage area, power train etc. Acceleration was measured on all 4 tyre hub simultaneously by acquisition system while moving on different road. These different acceleration load cases in the form of PSD were used as input in Ncode design life software to calculate the fatigue damage. The other input to Ncode software is modal stress result of FEA model. Modal stress results compute in Radioss software by applying the unit acceleration load to FEA modal. The result of this new method was compared with conventional method for fatigue damage calculation. The comparison between results of both methods shows that capability of new method to properly interpret the weak regions. The main disadvantage of this new method is that it is applicable only for linear FEA model.

Abdullah *et al.*[9]demonstrated that modal and frequency response analysis is an efficient tool to understand the dynamic behaviour of the component. Lower suspension arm of vehicle was considered for study .Natural frequencies also calculated experimentally using Impact hammer test to validate the Modal analysis results. There is difference in the natural frequencies calculated by numerically and experimentally. This is due to material properties used for Simulation. FEA Static Analysis was done for finding the critical strain areas. Results of the Frequency response analysis calculated will further use for future vibrational fatigue analysis.

Choi *et al.* [10]demonstrated a new method for the faster vibration endurance test in frequency domain for battery fixed bracket. This method uses Single DOF for experiment. The axis direction is selected on the basis of severity of damage in different axis. In this case Z axis shows maximum damage. The acceleration value used for test was acquired by using accelerometer on setup from actual different road types (expressway, national highway, unpaved road, City Street and local road). Acceleration signal from different road types was converted into PSD by FFT and these different road PSD values combined to make one overall PSD. Another method was six DOF methods in time domain for vibration test were also used. These methods were compared and show that single DOF accelerated test is more reliable and faster.

Khan *et al.*[11] demonstrates a method for fatigue life calculation by using RMS stresses and Basquin's relation. A component was selected for study is chassis mounted that is Auxiliary heater bracket. Abaqus was used for the simulation and FE model was created in Altair Hyper mesh. Initial FEA analysis shows that bracket not endured infinite life in 1 sigma level. Result shows maximum stress was 1022 MPa which was very higher than material yield strength (280 MPa). Redesign of Auxiliary Heater Bracket achieves infinite fatigue life in 1 sigma and 2 sigma level of confidence in FEA simulation.

Sontakke *et al.* [12]optimized the bracket and new model was proposed with reduce weight. The Bracket chosen is chassis mounted which supports the engine. This bracket suffers large intensity of vibration from engine. Harmonic analysis was performed to check the response of bracket under operating frequency range .The analysis result show that response acceleration value was under safe limit.

Yanping *et al.*[13] proposed a methodology for making the design of auto drive axle housing for fixed life cycle. The CAD model of axle housing was modified so that it represents the experimental mass and stiffness value. Fatigue life of auto drive axle was calculate by using miner cumulative theory and Steinberg three interval method based on stress The drive axle of truck was selected for this study. Natural frequencies are find by numerically and experimentally method. The difference in between natural frequencies calculated by two method show an error of range 0.25 % - 22.5 %. Mass of FEA model and actual drive axle is almost same. The equivalent FEA model was created which included truck drive axle, suspension system and vehicle body. Random vibration analysis was performed on equivalent system in ANSYS software to find the three interval stresses. The fatigue life result show that fatigue is generated in axle housing before 600,000 km when the vehicle moving at constant speed of 72 km/hr

Singh *et al.*[14] investigated the influence of rubber dampers on engine NVH (Noise ,Vibration and harshness) using Numerical analysis and experimental measurement .Use of Rubber dampers between fins of engine would help in reduce the high frequency radiated noise at higher speeds and reduce the fins vibration amplitude .But on the other hand it increases the overall cost and have high negative impact on environment .It also increases the engine temperature by 10% .To eliminate rubber dampers they proposed new design of engine ,which include ribs b/w fins. By modal analysis it was observed that fin local lower modes were the dominant modes of vibration. It was also found that amplitude of vibration of fins reduces at higher frequencies

For achieving the objectives, 4 iterations had been made in design of engine head. By design iterations, first mode natural frequency changed to 4137 Hz from 1470.3 Hz by adding ribs between the fins. There is also significant decrease in the vibration response at resonance from design 1 to design 4 as show by harmonic response analysis. So, it was concluded that the ribs b/w fins were the good alternative at the place of rubber dampers.

Loh *et al.*[15] performed the optimization of P-TAC motor bracket. Bracket is used to support motor and fan in air conditioning unit. The bracket is prone to vibration loads, so to check its design strength different analysis has been performed on bracket. Static analysis result of original bracket shows that static von-mises stress value is less than the yield stress but dynamic analysis of bracket under vibration environment shows that von-mises stress crosses the yield stress of material during external loading for 5 sec. Frequency response analysis was also performed to check at which natural frequencies response is maximum. Fatigue analysis of bracket demonstrate that original bracket design was failed under different fatigue life criteria like modified

Goodman, Soderberg and Gerber. Therefore, design modification was done in bracket by changing the radius, thickness and by adding diamond rib in bracket. These design points are changed five times to create different alternative design of bracket. Then these different design points are analysed by different analysis as done for original design and best design is select from them.

Raghavan *et al.*[16] suggested the approach for component design with complex functional requirement. This approach includes Static analysis and experimental durability testing. Linear static analysis would over predict the stresses in the component because of the non-liner loading and result in rejection of the design. So, that why non-linear static analysis was done to incorporate the outcome of loading beyond elastic limit. Non-linear Static analysis result shown that von-mises stress occurred in the legs of bracket is far belowthe tensile yield strength which means there were chances of mass optimization. By just changing the thickness of the legs 8% weight reduction was possible. This new bracket design also passes the required life of 400000 cycles at load checked by experimental durability testing.

Gang *et al.*[17] studied the characteristic of vibration of structure of Fiber Optical Gyroscope (FOG) bracket. To check the response of bracket in vibration environment, normal and side direction of the bracket was analysed till 2000 Hz by harmonic response analysis. Response Frequency curve show that no resonance condition occurred at first frequency in vertical direction but in side direction resonance occurred at fundamental frequency. For increasing the first natural frequency of bracket wall thickness is increased by 50% and height is decreased. Again harmonic response analysis was done on bracket and shown that longitudinal response of bracket to frequency excitation was decreased .So, overall response of bracket at resonance decreases.

Chang *et al.*[18] demonstrated that the initial design concept was very important in design stage of any component. Wrong initial design possibly will lead to higher cost of design cycle and long time to recover to the right direction. An Initial design concept was depends upon the designer experience. Compressor bracket was taken for analysis. Initially, to check the reliability of FEA model for structural analysis and modal analysis, FEA results were compared with experimental results and it shows the reliability of FEA model. Compressor bracket was optimized by topology optimization technique by using optimization code of Altair Engineering. After optimization bracket mass increases by 35 g, frequency in increased by 29 Hz and decreases the average stress to 40% against the initial concept design. It concluded that topology optimization is very effective tool with minimum mass increment.

Singh *et al*. [19] analysed the vibration level in Railway boggie and Truck during shipment of goods to various metropolitan cities in India. The standard way to represent the vibration level is by area under Power density curve (PSD) which is G_{rms} values. The comparison of India Rail and Truck vibration intensity with the standards (ASTM and ISTA) shows that vertical vibration level was more severe than existing standard. It also demonstrates that measured vibration level more severed in vertical direction than in lateral direction and least in longitudinal direction. Figure 2.1 shows the comparison of PSD for three orthogonal directions for railway boggie and Figure 2.2 shows the comparison of PSD for three orthogonal directions for heavy truck.

Yang *et al.* [20] studied the effect of road induce vibrations that is basically random in nature and harmonic vibration come from motor on mounting bracket. The effects (stresses and deformation) from these two forms of vibrations should be under safer limits. To ensure the performance of bracket it was simulated in virtual vibration environment provided by Abaqus software. FEA result shows that the response in terms of displacement and stresses resulting from either vibration analysis were far from damaging.

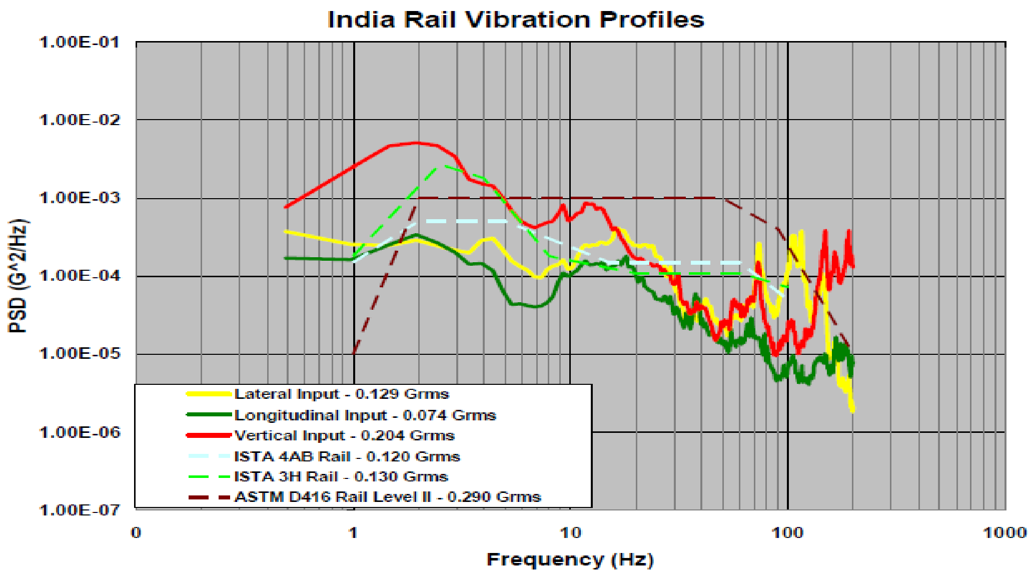


Figure 2.1: PSD plot for Rail Bogie in three orthogonal directions

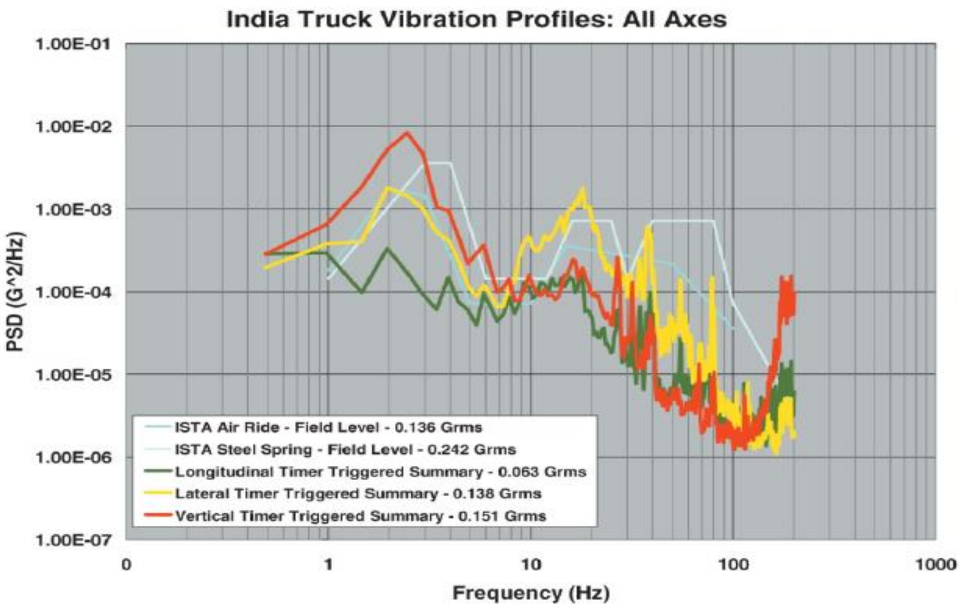


Figure 2.2: PSD plot for Truck in three orthogonal directions

Chapter 3

PROBLEM FORMULATION

3.1 Gaps in Literature

From the literature survey in the broad field of "designing of automotive components" mainly mounted brackets, it is noted that significant work has been done in the area of Finite element analysis (FEA) using static structural and dynamics analysis of automotive components. Some of the gaps have been found during literature survey are as follows:

1. Ample research is dedicated to the analysis of engine mounting bracket, suspension bracket. However, it is noted that less work has been reported in the area of static and dynamic analysis of chassis mounted brackets.
2. For most of the dynamic load investigations, mainly periodic excitation has been considered on the structural component. In limited studies, random vibration analysis of the mounting brackets of automotive vehicles is reported.
3. Many optimization studies are reported in the literature using static as well as periodic loading. Indeed, there is an ample scope of shape and weight optimization of automotive components using the actual road vibration data, which is rarely considered in the previous studies.
4. Previous studies are mainly focused on the static analysis, modal analysis and Frequency response analysis but transient response analysis of brackets is limited.
5. For the dynamic study, harmonic or periodic excitations generated from various rotating elements are considered for the loading on the structure, investigation using actual road vibration data is rare.

3.2 Objectives

By studying the gaps in the reviewed literature, following objectives have been formulated:

1. Formulation of a methodology for doing the dynamic analysis of chassis mounted bracket using commercial software under actual road vibration data

2. Measurement of real time road vibration signal on the supportive structure of the selected bracket by using data acquisition system.

3. Static structural, modal and transient response analysis of the selected automotive chassis mounting bracket using commercial software. .

4. To perform the Optimization of selected bracket for decreasing the mass of bracket.

3.3 Methodology for dynamic analysis of Bracket

The methodology used for the dynamic analysis of chassis mounted bracket is as shown in Figure 3.1:-

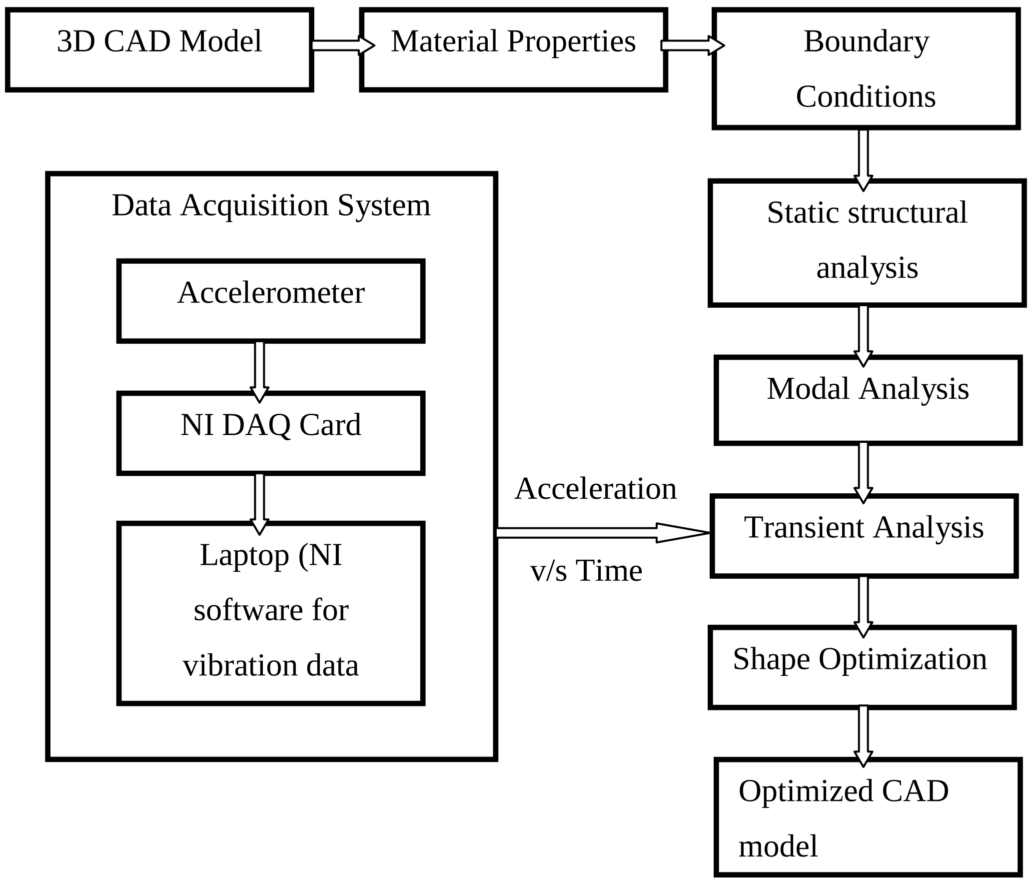


Figure 3.1: Flow chart of different steps for dynamic analysis of bracket

1) CAD modeling of the selected bracket in CAD tool (CREO software)
2) CAD model is imported in the CAE software (ANSYS) using the interface between these two software's
3) Pre-processing of the bracket in the ANSYS software
    a) Material properties to be defined
    b) Mesh generation
    c) Boundary conditions
4) Solution for Static structural analysis and Modal analysis
5) Post processing of the results in the ANSYS software
    a) Von Mises stresses, Total displacement, Natural frequency and normal mode shapes
6) Acquisition of vibration data in the form of Acceleration v/s time on bracket and on supporting structure of bracket by using data acquisition system.
    a) Post processing of Data
        I. Converted into the frequency domain by using FFT (Fast Fourier Transform) algorithm in MATLAB.
7) Transient analysis of bracket is performed in ANSYS.
8) Post processing of results
    a) Maximum Von-mises stress , Total displacement
9) Shape optimization of bracket in ANSYS software considering mass reduction as an objective function.

Chapter 4

DATA ACQUISITION SYSTEM

4.1 Procedure for acquiring the vibration data

The bracket considered in this work is mounted on the side member of chassis of the Eicher truck (Model No:-Turbo 11.10, 8 tonne capacity).Generally, brackets in a vehicles are affected by the road induced vibrations and by engine induced vibrations. The location of the selected bracket is at the rear side of the truck closed to rear suspension. As it is away from the engine, it is expected that it has more influence of road induced excitation and shock loads than the excitation induced from the engine unbalances. The vibration data from the road are acquired in the form of acceleration v/s time by using a data acquisition system. Later this vibration data are used to do the transient dynamic analysis of the bracket. The following procedure is used to acquire the vibration signal:-

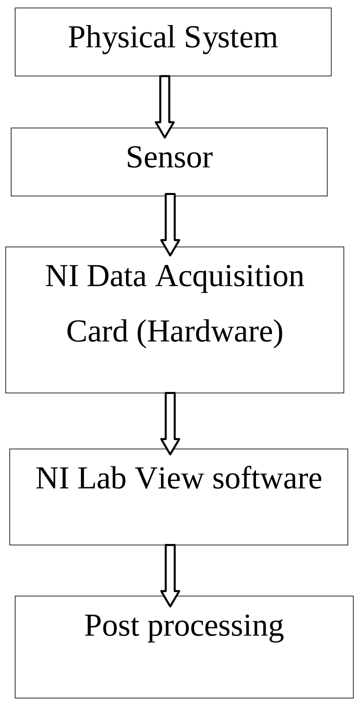


Figure 4.1: Flow chart for different steps of Data acquisition system

The different constituents of the flowchart shown in Figure 4.1 are briefly explained below

1. Physical system

A chassis mounting bracket is considered for analysis. Figure 4.2 below shows the chassis mounted bracket.

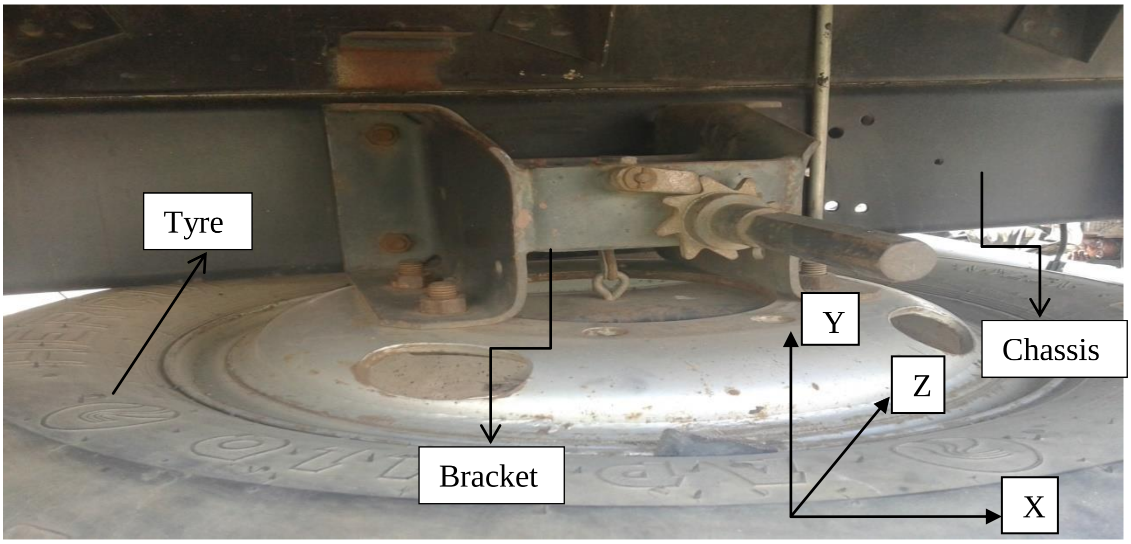


Figure 4.2: Chassis Mounted Bracket supporting the spare tyre

(a) **(b)**

Figure 4.3: Accelerometer Position for Vibration Measurement in (a) Longitudinal Direction (b) Vertical Direction

2. Sensor

Sensor is a device that uses to measures a physical quantity like displacement, load, pressure, temperature etc. In the present study, acceleration of a vibrating system is measured. Sensor converts the measured quantity in into a measurable electrical signal. This output electrical signal can be in the form of current, voltage or resistance that varies over time, depending upon type of sensor. In the present system vibration is measured with the help of accelerometer sensor. One of the accelerometer positions on the chassis mounted bracket is shown in figure 4.3. Bruel and Kjaer Piezoelectric accelerometer Type 4533-B [21] is used in this study for acquiring vibration signal.

3. NI Data acquisition card: -

It acts as an interface between sensor and computer. It is used to digitize the signal coming from sensor so that computer can interpret them. Main components of the DAQ are signal conditioner, analog to digital convertor and computer bus. Output of the sensor is connected to one of the five input ports of NI DAQ card. It further sends the conditioned signal to computer for further processing. DAQ card used in this study is NI USB-4432 [22]

4. NI Lab view software:-

It is programmable software used to control the operation of DAQ device and visualizing, storing the measured results on computer. The following procedure is used in the NI Lab View software to interpret the signal:-

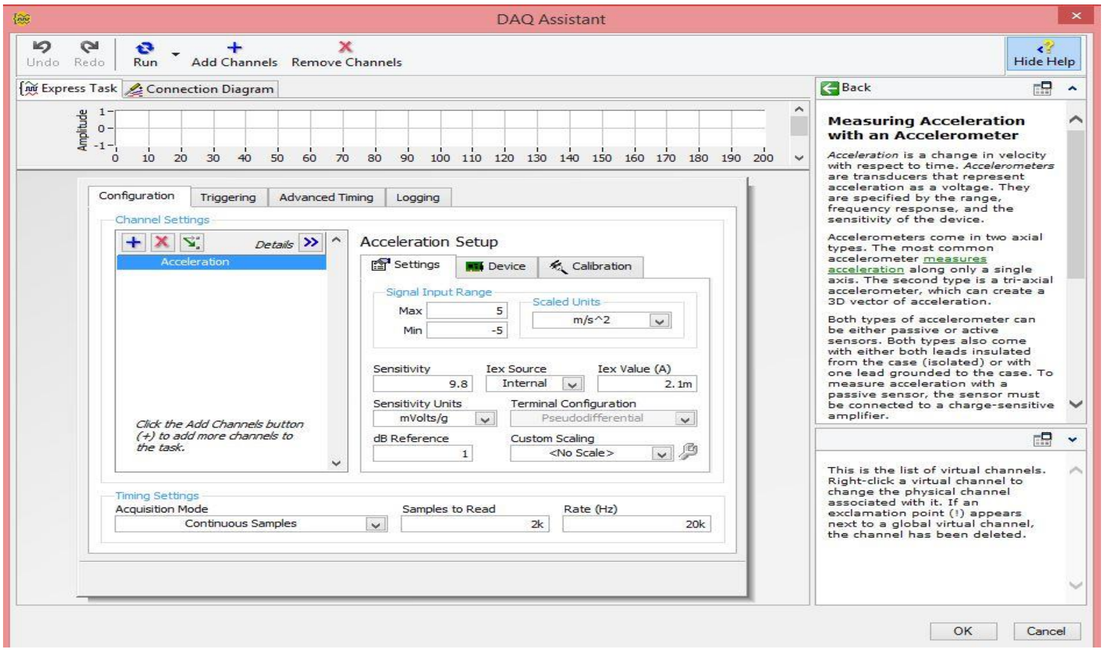


Figure 4.4: DAQ Assistant Window of Lab View Software

a) The first step is to configure the DAQ card and accelerometer using the DAQ assistant window shown in Figure 4.4. It is used to configure the port of DAQ card in which the accelerometer is connected. Accelerometer properties like sensitivity, sensitivity units, scaled units etc. are also defined in this window.

b) Next step is to create the user interface and block diagram code by using the front panel window and block diagram window respectively of Lab View Software. The front panel consist of controls, knobs, display screens etc. which is basically used to define the parameters, giving instructions to control the program and display the results. To create the code in Lab view, inbuilt functions are used. Different functions are connected by virtual wires. Sequence of functions defines the flow of data in wires. Fig 4.5 and Fig 4.6 shows the front panel and block diagram window, which is used in this study to acquire the vibration signal.

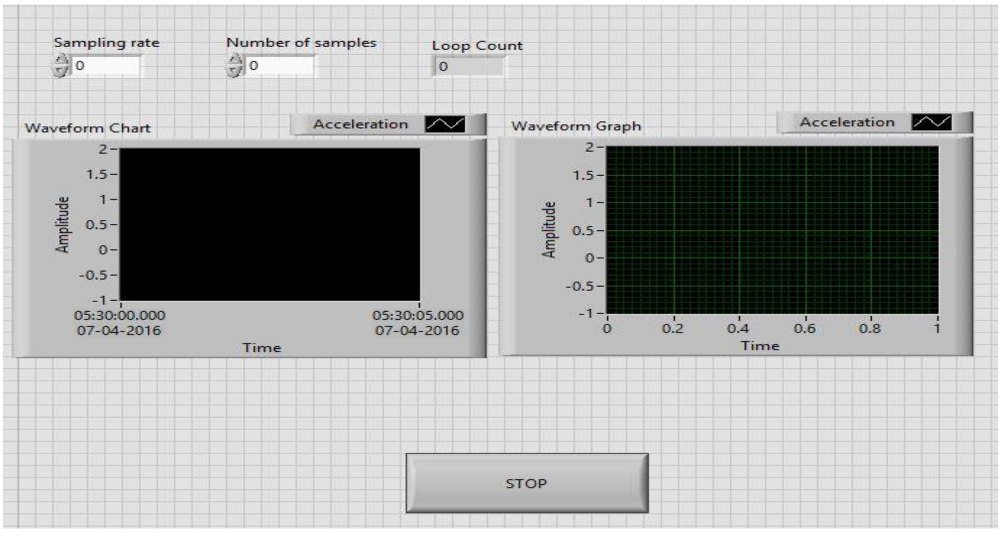


Figure 4.5: Front Panel Window in Lab View Software

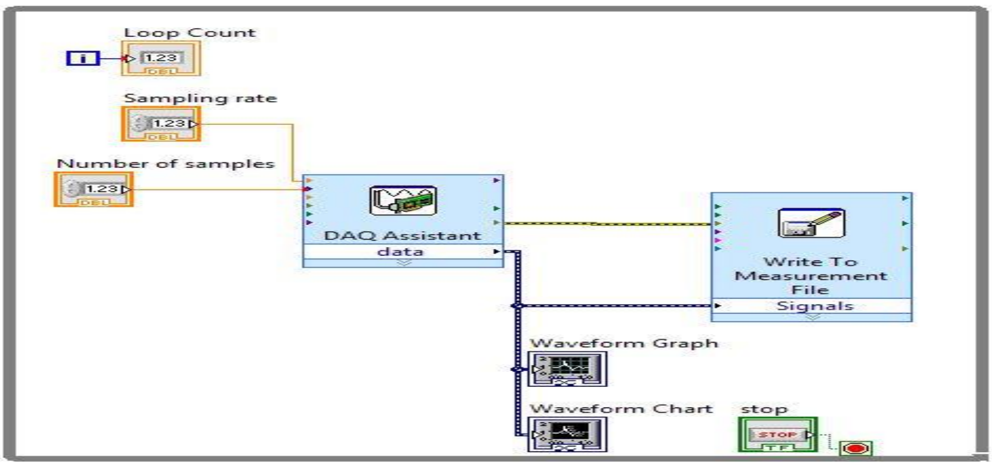


Figure 4.6: Block Diagram Window in Lab View Software

4.2 Parameters required during experiment:-

Following two parameters are used as input in front panel window.

Sampling Rate: - 15000 Hz

Total record time for signal: - 40 sec

4.3 Results and Discussions

A setup was created for acquiring the real time vibration data from the mounting position of Chassis mounted bracket. The setup was used in a moving truck. Real time vibration data was measured in only two directions vertical (y) and longitudinal (x) as shown in Figure 4.3. These results were saved on computer by using the Lab View Software. Data was collected from different types of roads and post processed using in-house developed code in MATLAB. The data are compared mutually on the basis of root mean square values (RMS) and power spectral density values (PSD) using developed code.

4.3.1 Real Time Vibration Acquisition Result

1) Data acquisition from vehicle moving on Inter-City Road

The road data is measured for an inter-city road**.** The data is measured for a distance of 0.6 Km approximately. The measurement is repeated for 4 times. Figure 4.7 shows time history of the acceleration of one of the most severe data (among the measured 4 samples) measured in the vertical direction on the chassis. The data is captured for 40 seconds at a sampling rate 15k. During the measurement, the speed of the vehicle is maintained between 40 kmph to 50 kmph. The accelerometer position for vibration measurement in vertical direction is shown in Figure 4.3(b). It is observed that the nature of the acceleration data is non-deterministic random signal. Figure 4.8 shows the PSD of the measured data (Figure 4.7). From the PSD plot, it is observed that the signal has a notable strength in the lower frequency range (less than 15 Hz). [19], [23] have also found a dominating frequency within 1 to 15 Hz in the PSD of measured road data. It also observed that maximum PSD value occurred in 3-5 Hz range. Many researchers have described that highest PSD value in the range 3-5 Hz is due to truck suspension response [23]

a) Vertical Direction

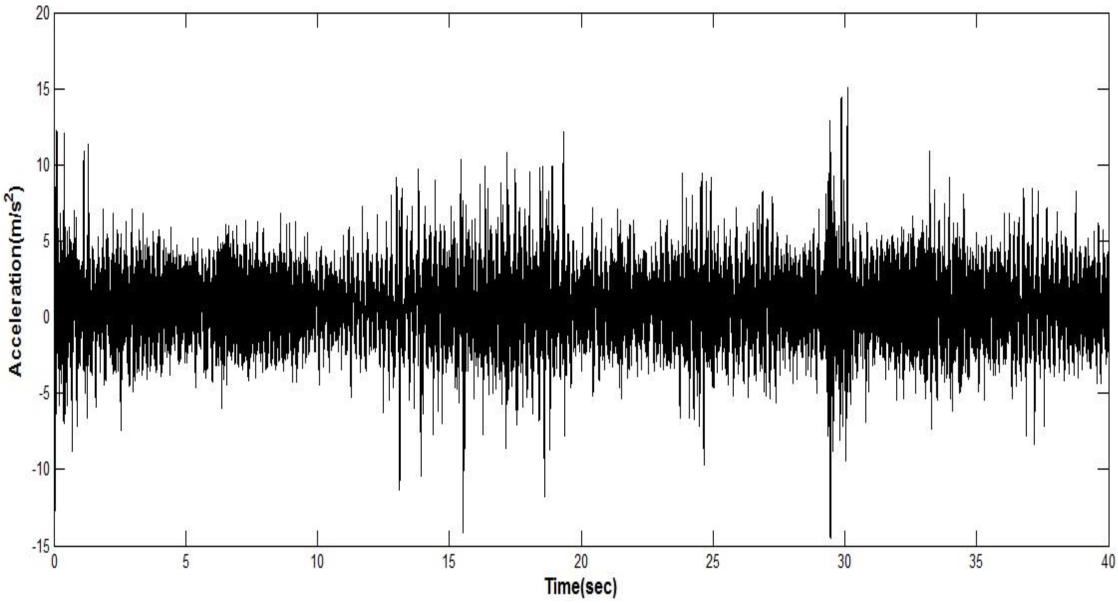


Figure 4.7: Time History of the acceleration in vertical direction for Inter-City Road

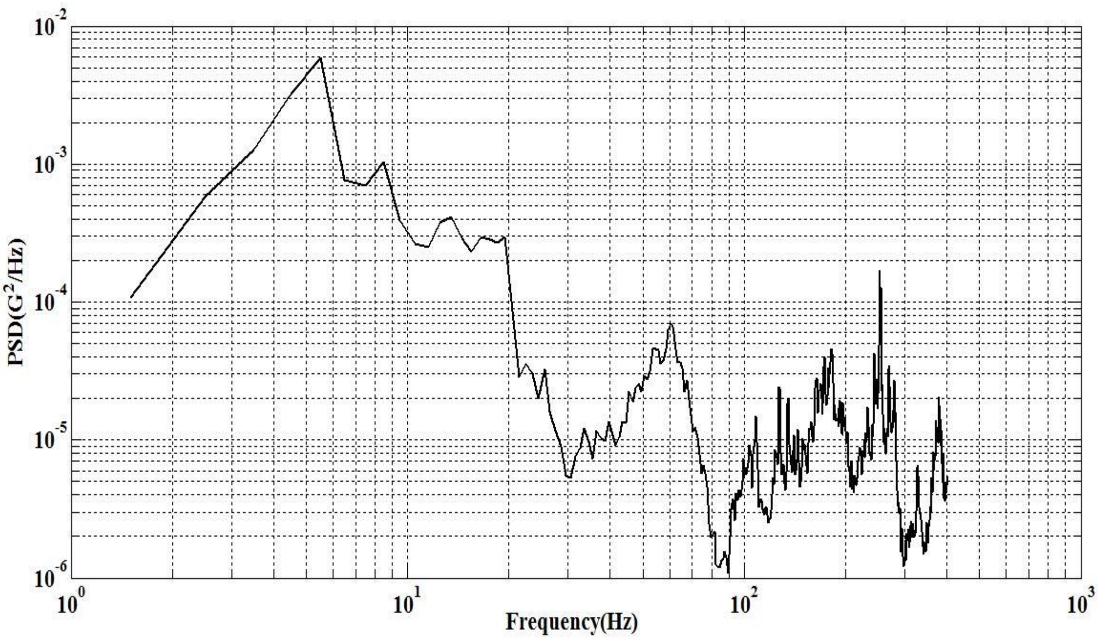


Figure 4.8: PSD profile for Inter-City road in vertical direction

(b) Longitudinal Direction

The vibration data is acquired on bracket in longitudinal direction for 40 sec with the speed of truck is maintained between 40 kmph to 50 kmph. The accelerometer position for vibration measurement in longitudinal direction is shown in Figure 4.3(a). Figure 4.9 shows time history of the acceleration of one of the most severe data (among the measured 4 samples) measured in the longitudinal direction at the root of the bracket. Figure 4.10 shows the PSD of the measured acceleration data (Figure 4.9).The PSD graph for longitudinal direction shows the notable strength in the lower frequency range (1-15 Hz) but the value of PSD is much less as compared to PSD values for vertical direction.

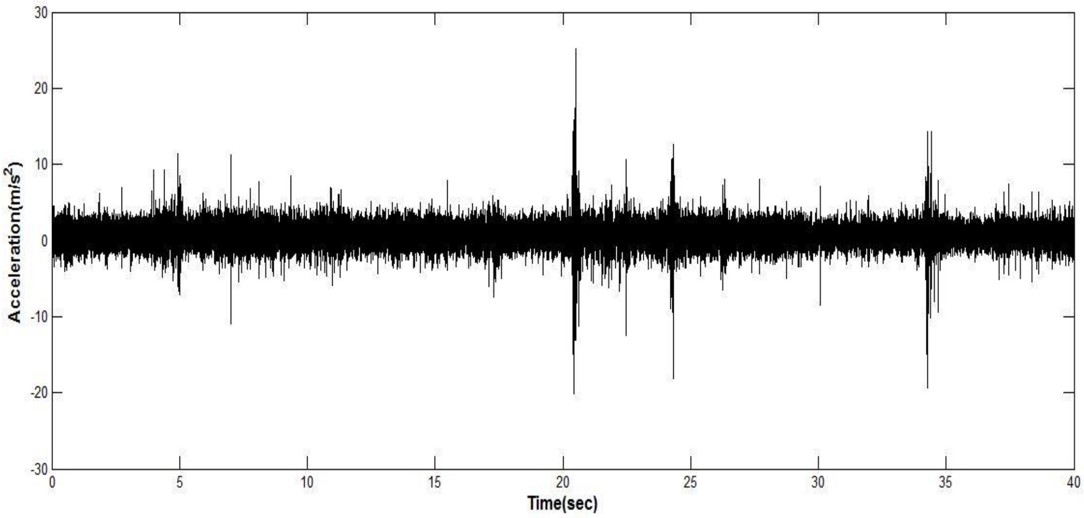


Figure 4.9: Time History of the acceleration in longitudinal direction for Inter-City road

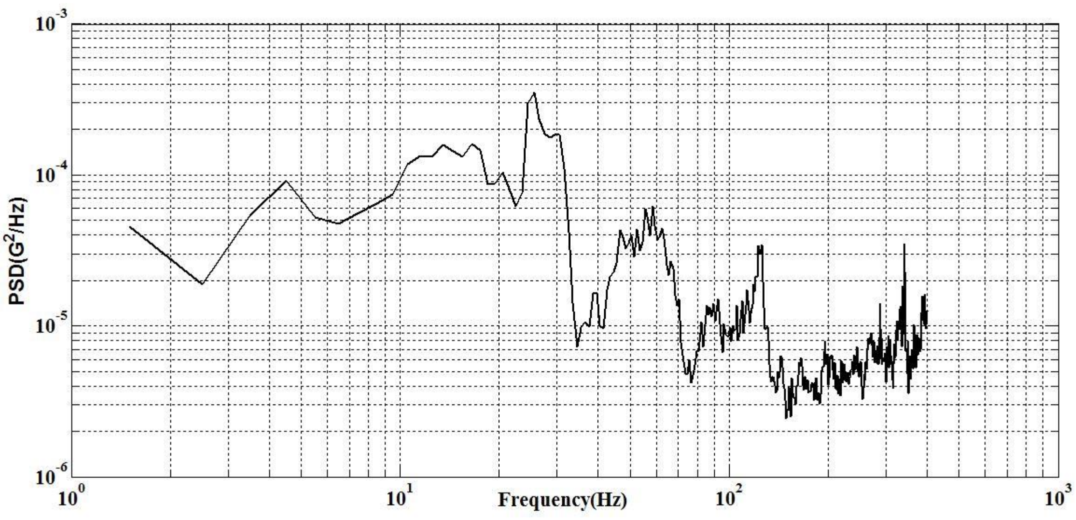


Figure 4.10: PSD profile for Inter-City road in longitudinal direction

2) Data acquisition from vehicle moving on Rural road

The acceleration data is measured for a rural road**.** The road condition of rural road is bad as compared to Inter-city road due to pit holes. Same process is repeated for rural road to acquire the vibration data in vertical direction and longitudinal direction as discusses earlier for inter-city road .But the speed of truck is maintained between 20-30 kmph due to condition of road. Figure 4.11 and Figure 4.13 shows time history of the acceleration in vertical and longitudinal direction respectively .It is observed that the nature of the acceleration data is non-deterministic random signal. Figure 4.12 and Figure 4.14 shows the PSD of the measured data (Figure 4.11) and (Figure 4.13) respectively. PSD curve for vertical direction of rural road shows the same behaviour as compared to PSD for Inter-City road in vertical direction. But the PSD value for rural road in vertical direction is little high due to bad road condition.

a) Vertical Direction

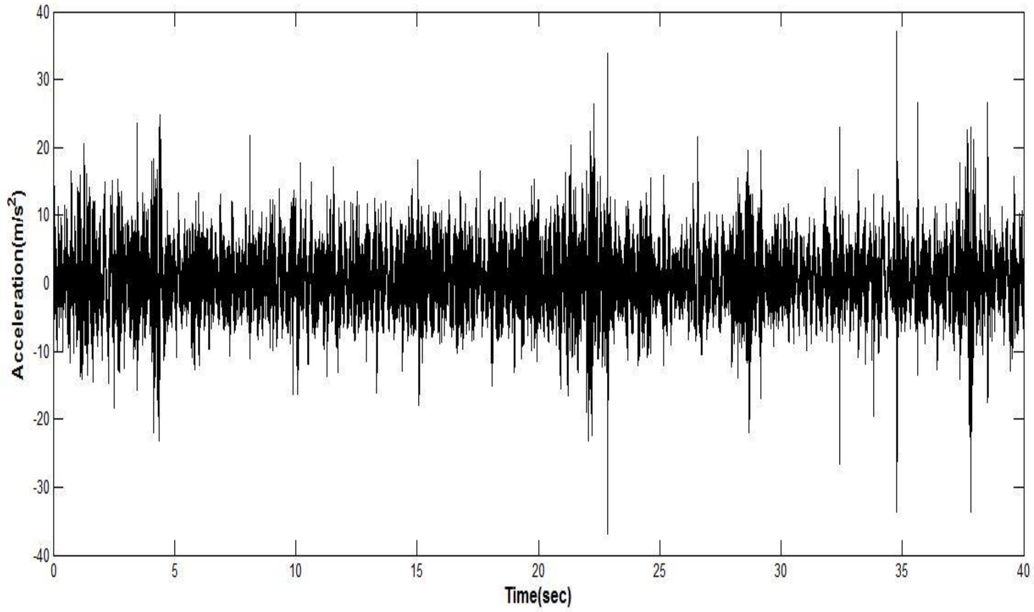


Figure 4.11: Time History of the acceleration in vertical direction for Rural road

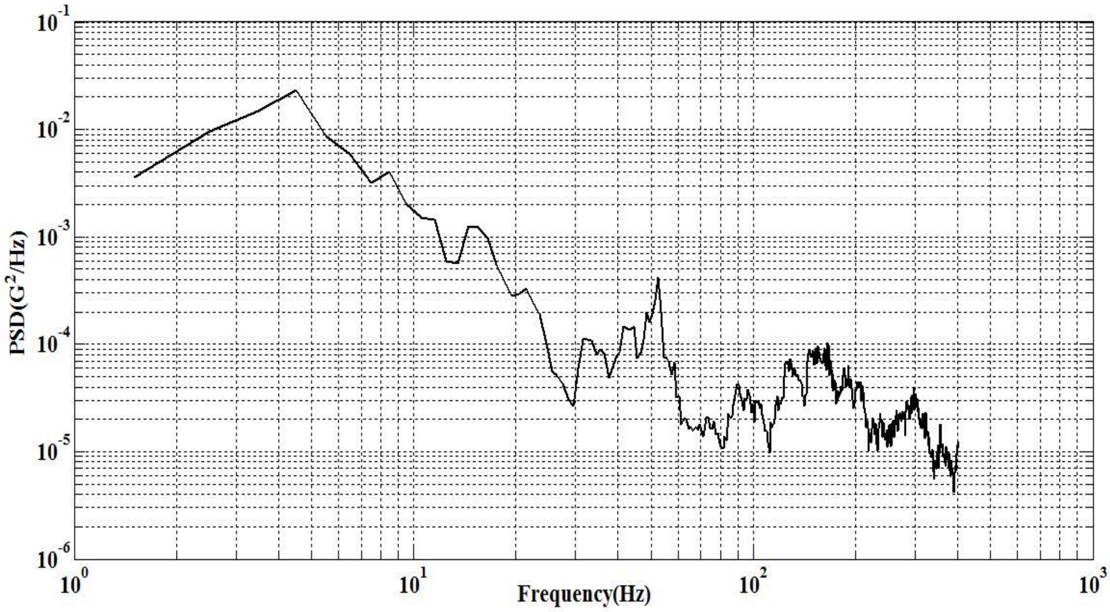


Figure 4.12: PSD profile for rural road in vertical direction

b) Longitudinal direction

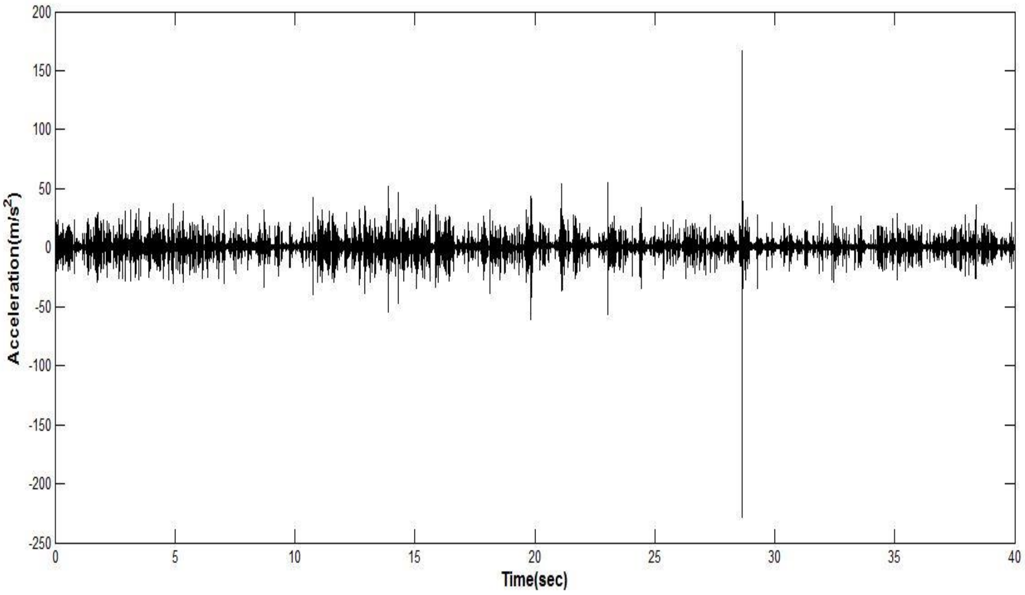


Figure 4.13: Time History of the acceleration measured in longitudinal direction for rural road

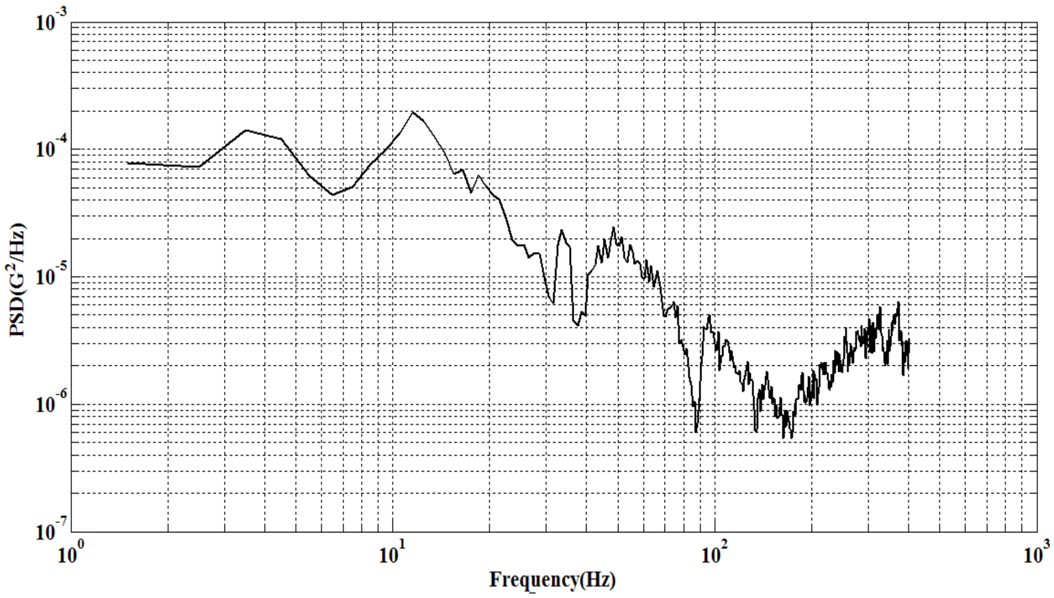


Figure 4.14: PSD profile for rural road in longitudinal direction

4.3.2 Root Mean Square (RMS) value for different roads

Table 4.1: RMS Values for Rural road and Inter-City Road

Vibration Signal RMS Values		
	Rural	**State Highway**
Longitudinal (x-axis)	0.3646	0.1862
Vertical (y-axis)	0.3714	0.1933

Table 4.1 represent the overall RMS value for two different roads. Vertical axis has the highest RMS value compared to other directions .The PSD graph of vertical direction also shows more power per frequency as compared to other axis. So, vertical direction has more vibration intensity. Therefore for transient dynamic analysis of bracket, only acceleration data in vertical axis is used to find the stresses.

PSD profiles also shows that maximum response occurred at smaller frequencies less than 30 Hz. This is due to bracket mounting location which is at rear of truck chassis. At rear location it is mostly affected by the lower frequency road induced vibrations.

Chapter 5

ANALYSIS OF BRACKET

5.1 Analysis of Chassis Mounted Bracket

A chassis mounted bracket of Eicher Truck (Model No:-Turbo 11.10) was selected for this study. The main purpose of the bracket is to support the spare tyre, which is very heavy and having a weight of 80 Kg approximately. The bracket has to withstand against the dead load from tyre and vibration load from road. There are also shock loads due to the impacts generated from pit holes while the vehicle moves on the road. Base of the bracket is fixed to chassis by fasteners and the tyre is mounted on the other end. The bracket was analysed in FEA software ANSYS to study the effects of static and dynamic loads. Fig 5.1 shows the position of chassis mounted bracket.

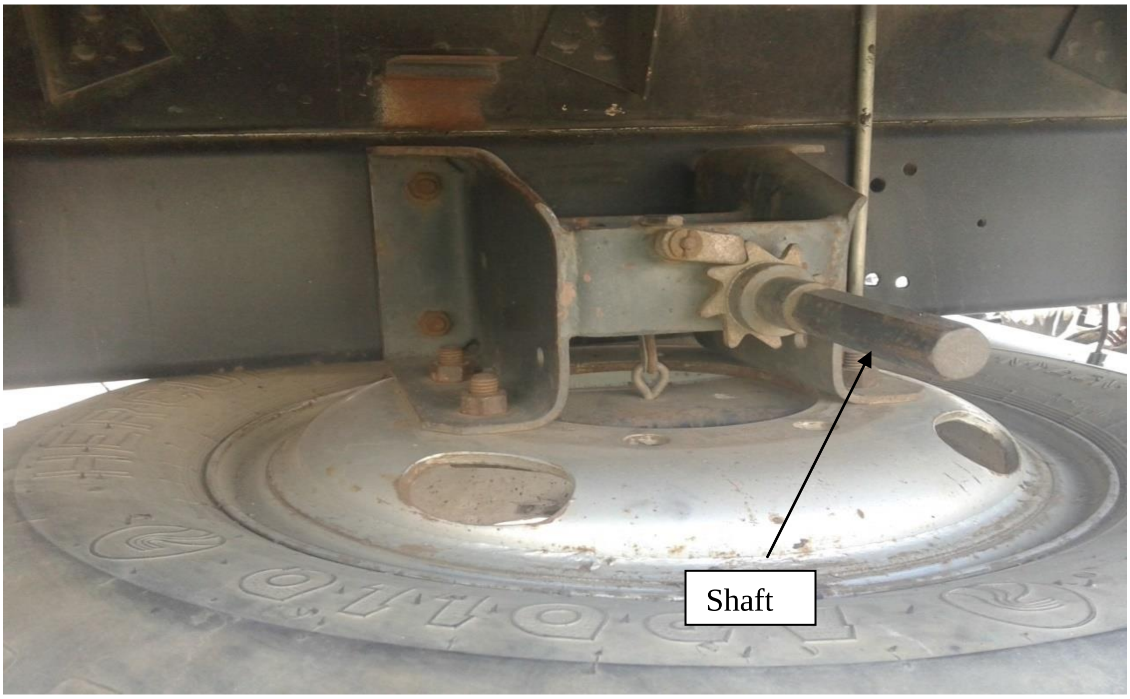


Figure 5.1: Chassis mounted bracket

For doing the analysis of bracket in ANSYS, CAD model of bracket is required. CAD model was created with the help of CREO software. But the drawing or dimensions of bracket was not available. So, reverse engineering was done on the bracket to find the dimensions.

5.2 Measurement of Bracket Dimension

A bracket is purchase and its dimensions are measured using Vernier calliper, height gauge, depth gauge, etc. The following figure show the bracket dimensions in three orthographic views.

.

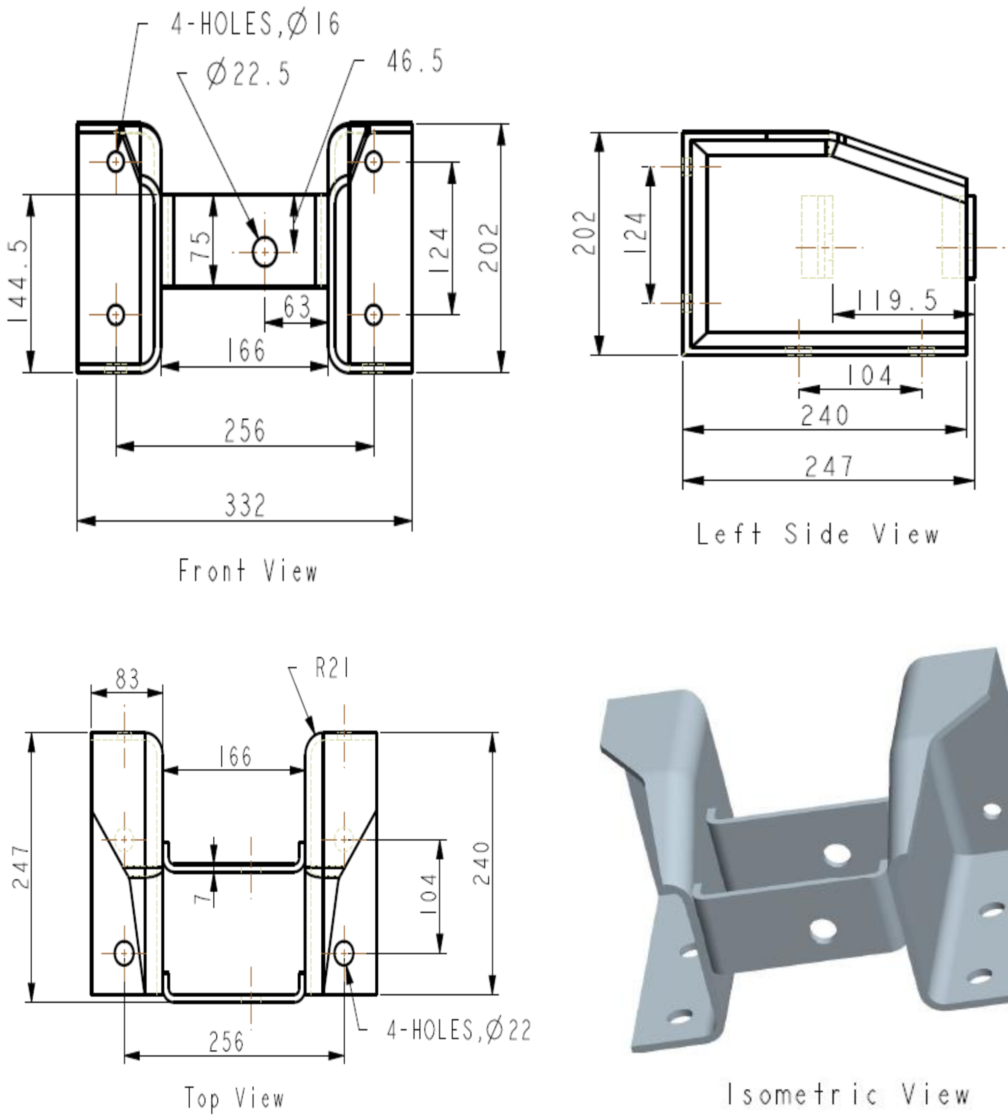


Figure 5.2: Chassis mounted bracket Dimensions (mm)

5.3 FEA CAD model

A 3D CAD bracket was modelled in CREO software by using reverse engineered dimensions measured in previous section. To properly imitating the actual dynamic conditions in virtual environment (FEA software), the other parts as chassis frame (C-shape), studs, washers etc. were also modelled. An assembly shown in the Figure 5.3 was created using CREO software.

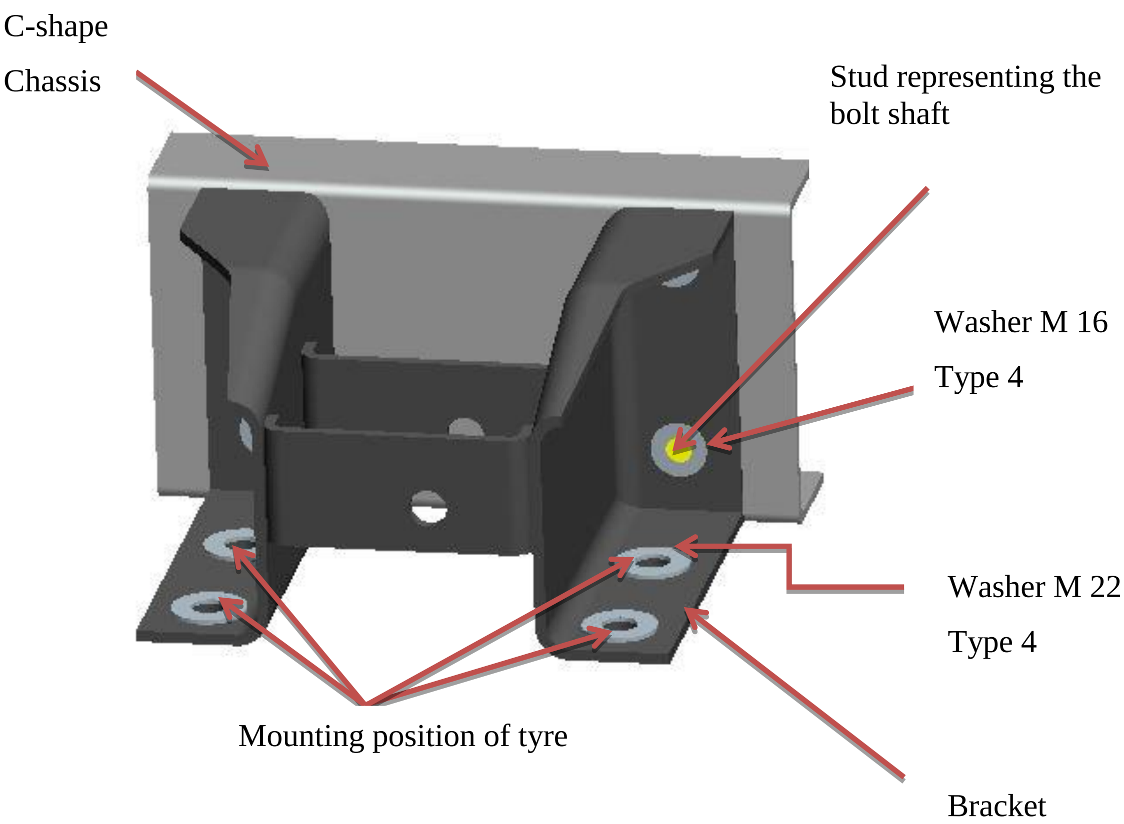


Figure 5.3: Component Assembly for FEA analysis

The C-chassis component was modeled on the basis of dimensions available from literature [24]. The dimension of cross section of C shape chassis is 210 X 76 X 6 mm. Washer and stud dimensions are taken from standards available in Machine drawing textbook [25]. The bracket has four mounting holes for tyre and four holes for mounting bracket to side member of chassis. Bracket and chassis has throughout uniform thickness of 7 mm and 6mm respectively.

5.4 Material Properties

A spectrometer was used to find the chemical composition of bracket. A small rectangular test specimen was cut from bracket and used in spectrometer machine. An experiment was performed three times and average was taken for composition.

Table 5.1: Results of spectroscopy for material composition

Element	Exp1	Exp2	Exp3	Average
Fe %	98.5	98.5	98.5	**98.5**
C %	0.137	0.124	0.132	**0.131**
Si %	0.0305	0.0393	0.0379	**0.0359**
Mn %	1.16	1.17	1.19	**1.17**
P %	0.0344	0.0269	0.0314	**0.0309**
S %	0.023	0.0226	0.0255	**0.0237**
Cr %	< 0.0030	<0.0030	<0.0030	**<0.0030**
Mo %	< 0.0050	<0.0050	<0.0050	**<0.0050**
Ni %	<0.0050	<0.0050	<0.0050	**<0.0050**
Al %	< 0.0010	<0.0010	<0.0010	**<0.0010**
Co %	0.0044	0.0052	0.0049	**0.0048**
Cu %	0.0054	0.0059	0.0054	**0.0056**
Nb %	0.0027	0.0031	0.0038	**0.0032**
Ti %	<0.0020	<0.0020	<0.0020	**<0.0020**
V %	< 0.0020	<0.0020	< 0.0020	**<0.0020**
Pb %	<0.0150	<0.0150	<0.0150	**<0.0150**
Sn %	<0.0020	<0.0020	<0.0020	**<0.0250**
B %	<0.0010	<0.0010	<0.0010	**<0.0020**
Ca %	0.002	0.004	0.0004	**0.0003**
Zr %	< 0.0020	<0.0020	<0.0020	**<0.0020**
As %	< 0.0050	<0.0050	<0.0050	**<0.0050**

On the basis of composition, material was found to be a grade equivalent to steel. So, for Bracket structural steel was chose as a material. A structural steel is commonly used material in brackets manufacturing with less cost and better forming property. Structural Steel properties given in Table 5.2

Table 5.2: Properties of structural steel

Property	Value
Density	7082 kg/m^3
Poisson's Ratio	0.30
Elastic Modulus	2.0E11
Tensile Strength	460 MPa
Yield Strength	250 MPa

For Chassis a preferred material Structural Steel Grade St52 [26] was selected .Its properties are given below in Table 5.3

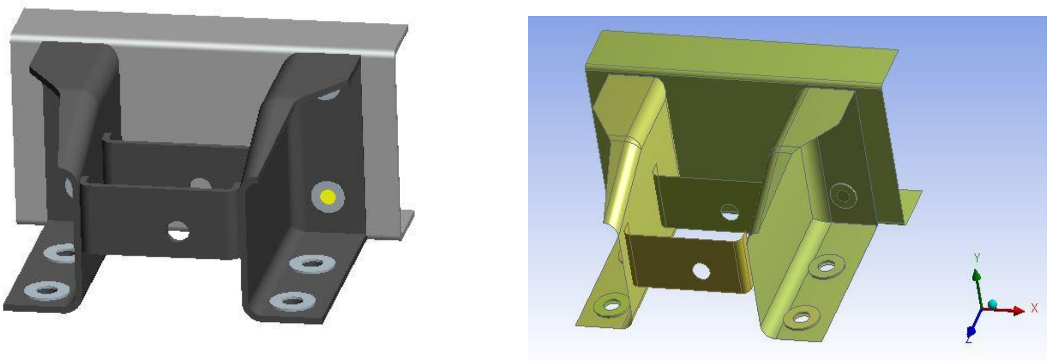
Table 5.3: Properties of structural steel Grade St52 [26]

Property	Value
Density	7800 kg/m^3
Poisson's Ratio	0.30
Elastic Modulus	2.1E11
Tensile Strength	520MPa
Yield Strength	360MPa

5.5 Connection definition and Mesh Generation

A solid model created in CREO software was imported to ANSYS software with the help of interface between software. By using CAD interface manager of ANSYS software, CREO software is connected to ANSYS. By this there is no need of converting CREO file format into neutral format. Before the mesh generation, following steps were performed in ANSYS:-

a) Bracket body and chassis structural component have uniform thickness, so it can be converted into mid surface at the middle of cross-section. Mid surface was created by using mid surface command in DesignModeler module of ANSYS. Like as shown in Figure 5.4 bracket is consist of 4 cross-sections and they one by one converted into mid surface. Figure 5.4 (b) shows the extracted surface model. Extracted surface was meshed by 2D shell elements or thin as comparison to 3D elements for solid Bracket and chassis. By this number of elements and nodes reduced drastically for same element size which decreases the computational time. Stud and washer were meshed by 3d elements.



(a) **(b)**

Figure 5.4: (a) shows the solid model before mid-surface generation and (b) solid model after mid surface generation

b) To define relation between parts of subassembly, connections have to define between different parts bodies like washer, chassis, bracket and stud. So that it can behave as like they are in an actual physical condition. Table 5.3 shows the connection type between different parts.

Table 5.4: Connection type between different parts

Parts	Connection Type	No. of connections
Washer M16 -Stud	No-separation	8
Bracket-Stud	No-separation	4
Washer M22 - Bracket	Bonded	4
Washer M16- Bracket	Bonded	4
Chassis-Stud	No-separation	4
Chassis- Washer M16	Bonded	4

c) A simplification of Bolt-nut-washer subassembly is done by representing it by washer-stud part. Stud part represented the bolt shank and washer represents the area of bracket and chassis affected by the washer in actual connection of nut bolt assembly. This simplification will mimic the actual bolt-nut-washer condition. This Bolted assemblies is used to mount bracket to chassis. Meshing of the full assembly is shown in the Figure 5.5. Accuracy or quality of mesh is defined by following parameters:-

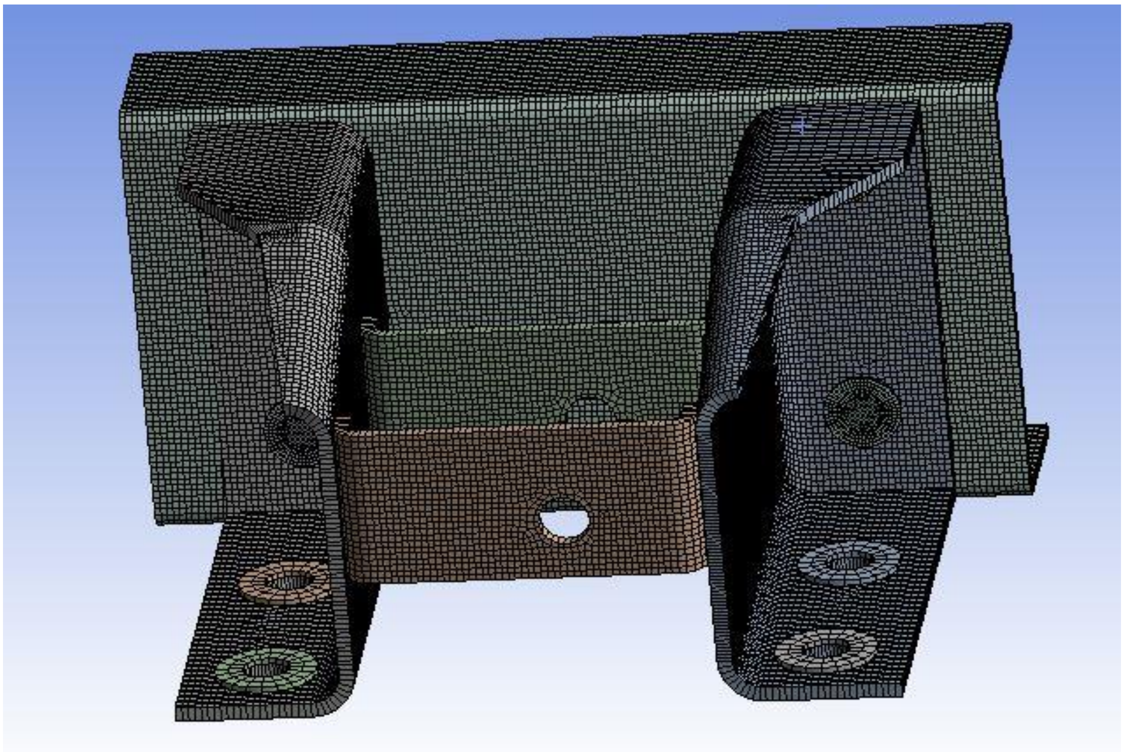


Figure 5.5: Discretized model of Bracket Assembly

i. **Aspect Ratio:**

It compares the edge length of element. The ideal shape for quad element is square with all angles 90 degree. It is defined as a ratio between longest element edge length to shortest element edge length. So its ideal value is equal to 1 and acceptable < 5.

ii. **Jacobian ratio:**

To check effectiveness of transformation of elements to local co-ordinate system from global coordinate system. Transformation is done to reduce the computation time during processing. Transformation is done for every element and coordinate system is placed at centroid

Ideal value = 1

Acceptable > 0.6

iii. **Skewness:**

It is defined as 90°-α

For quad element, α is a minimum angle between the two lines joining the opposite sides mid nodes.

Ideal value =0°

Acceptable < 45°

In ANSYS, these parameters value are calculated for every element and then average values are calculated by taking values for all elements. In Table 5.5 average value are shown.

Table 5.5: Values of Mesh Quality Parameter

Parameter	Value
Jacobian Ratio	1.0774
Aspect Ratio	1.1585
Skewness	0.08°

5.6 Static structural analysis of bracket

Static structural analysis was mainly done to find out the stresses and deformation values due to time independent loads and self-weight. In this study dead weight of tyre is the main external load on bracket .For doing the static analysis, boundary conditions are defined first and then static analysis was performed. Static structural analysis has been performed in ANSYS software. Figure 5.6 shows the boundary conditions which included forces, inertial load and support.

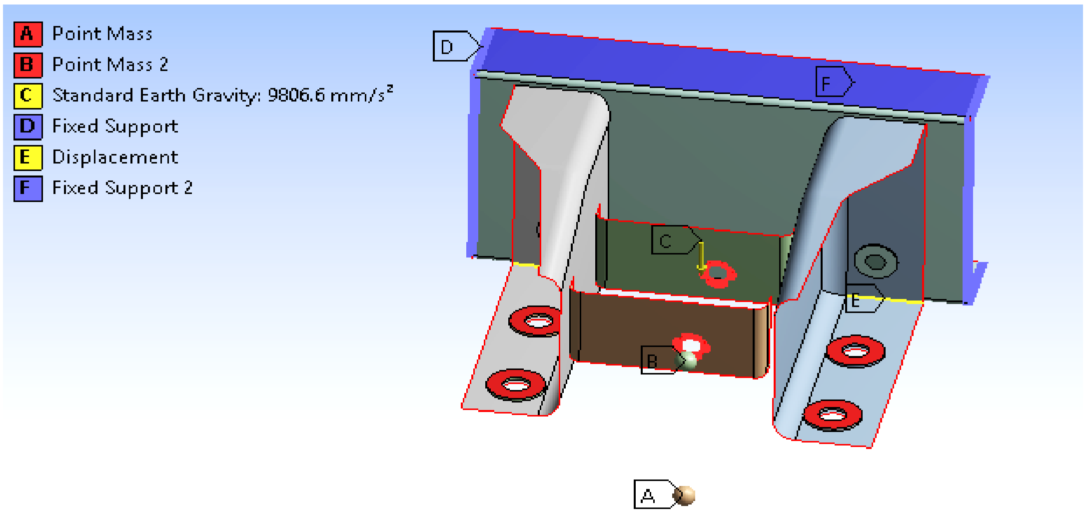


Figure 5.6: Boundary conditions for the Bracket Assembly

As the focus of static analysis it to find stresses and therefore, to simplify the analysis, non-critical parts are modelled as a point mass at CG.

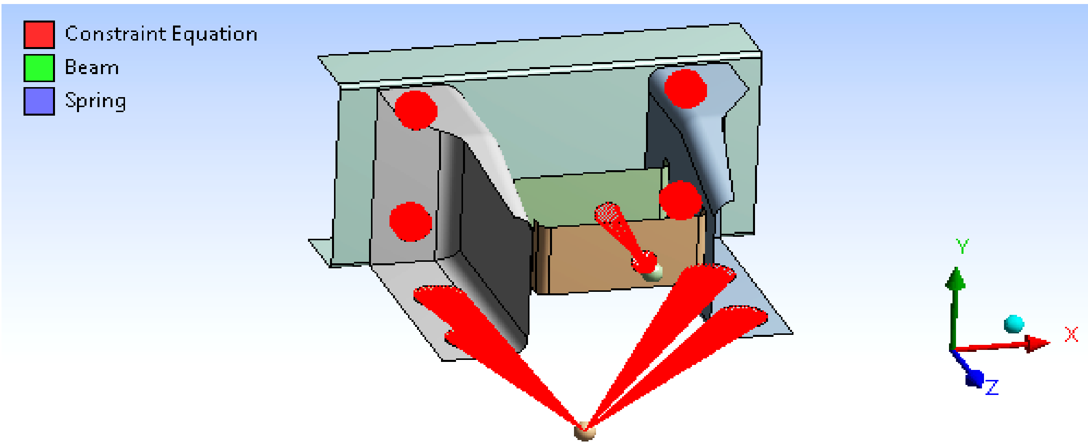


Figure 5.7: Connection of Point mass to their respective position

Point Mass represents the dead load mass of the tyre mounted on the bracket and its value is 80 Kg. The tyre is mounted on bracket with the help of 4 bolt connection as shown in Figure 5.1. Therefore, point mass A is remotely attached to four washer faces as shown in Figure 5.7. Point Mass 2 represents the Shaft mass and the location of application of point mass is at CG of Shaft. To find its CG position, CAD model of Shaft is created. Shaft position in bracket assembly is shown in Figure 5.1. The side faces and upper face of chassis are fixed by using fixed support boundary condition display as constraint D and F in Figure 5.6 .Constraint C defined the standard earth gravity in vertical direction which is used to include the effect of self-weight of bracket and to apply inertial force due to point masses at their specific position.

After defining the boundary conditions, static analysis was performed. The Figure 5.8 and Figure 5.9 show the results obtained after static analysis.

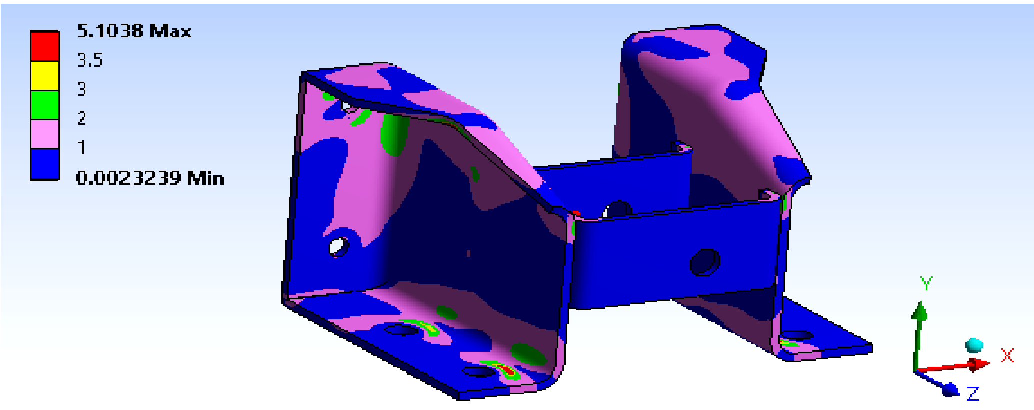


Figure 5.8: Von Mises Stress Distribution in Bracket (in MPa)

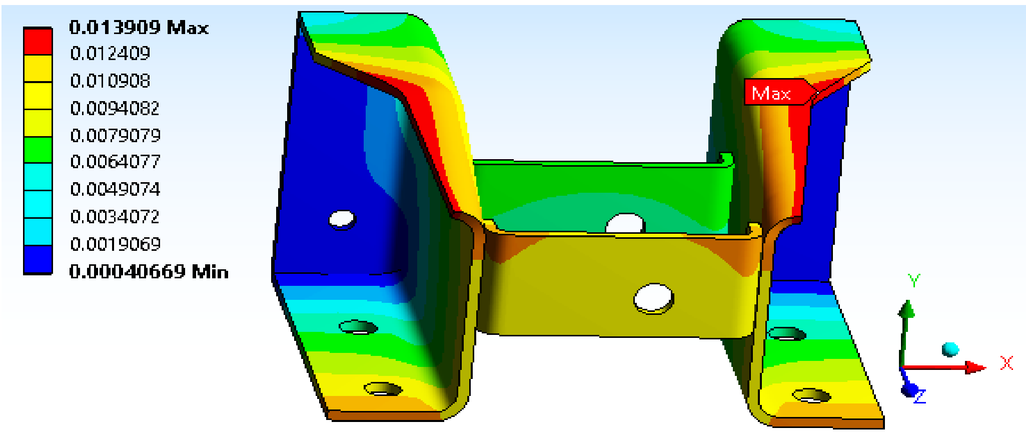


Figure 5.9: Total Deformation Distribution in Bracket (in mm)

The static analysis results show that the maximum Von Mises stress value is far away from Yield strength of material which is 250 Mpa. The maximum deformation value is also too less for the actual bracket design. These results conclude that the bracket is over designed.

5.7 Dynamic analysis of Chassis Mounted Bracket

Dynamic analysis of bracket assembly was performed to find out the following things.

1) Vibration parameter like natural frequencies and mode shape.
2) Response of bracket due to external excitation load (Vibration load).

The result of modal analysis was used further in transient analysis.

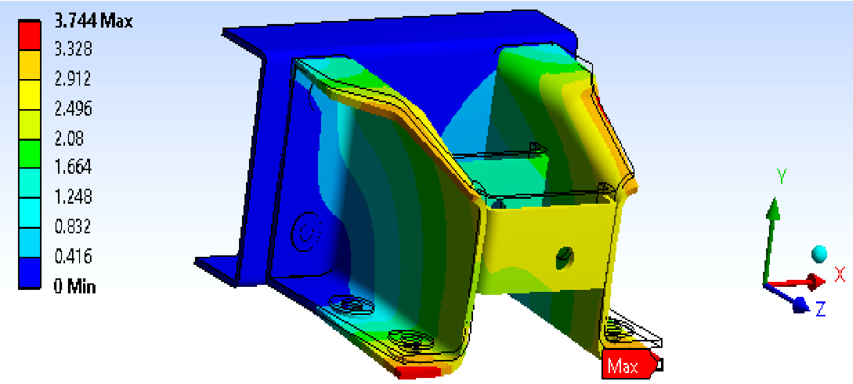
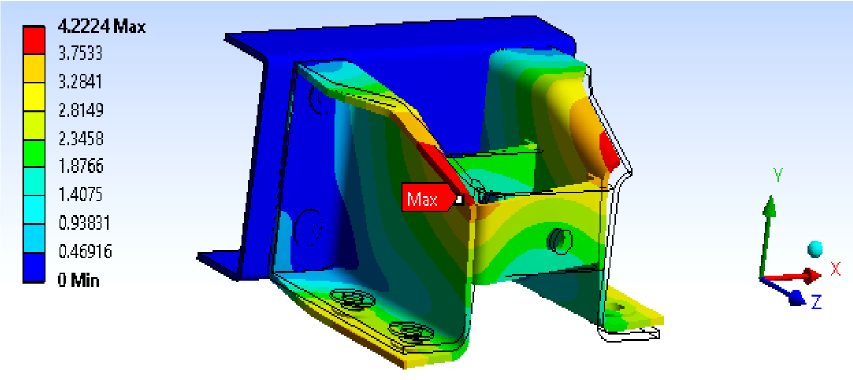
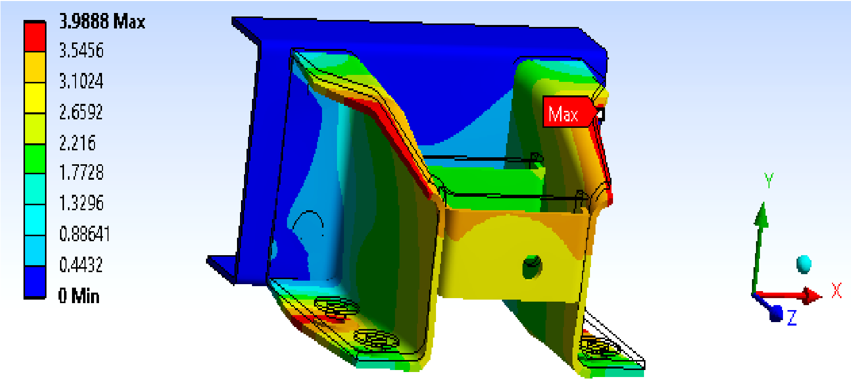
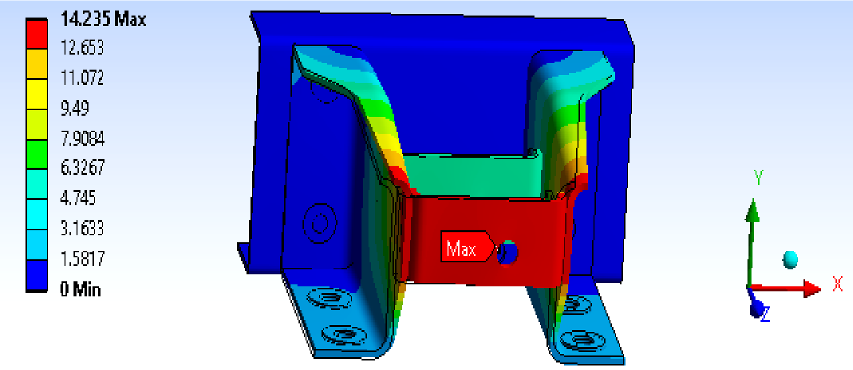
5.7.1 Modal Analysis

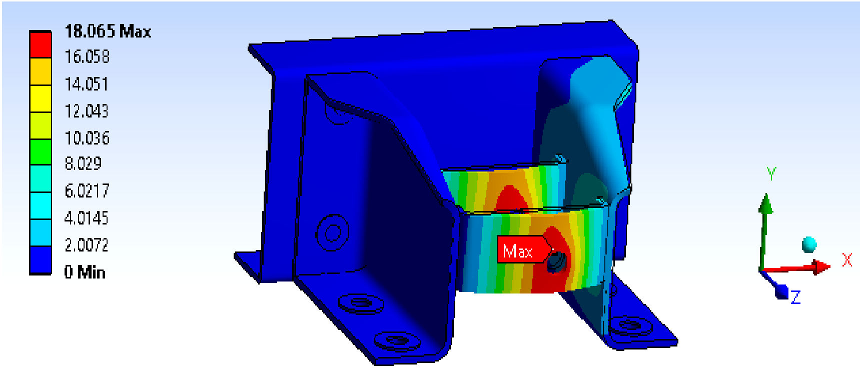
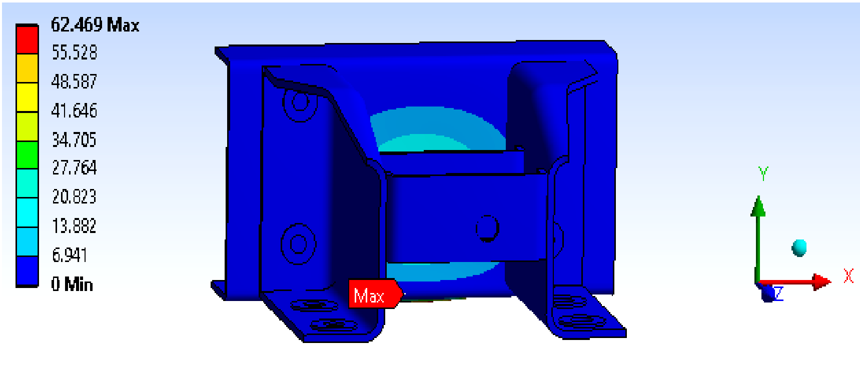
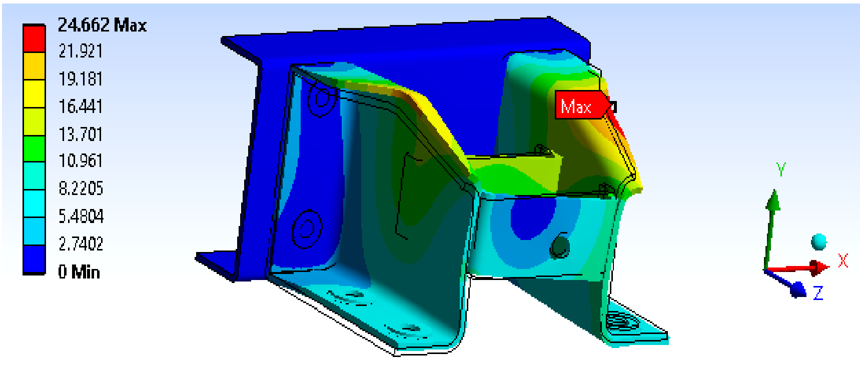
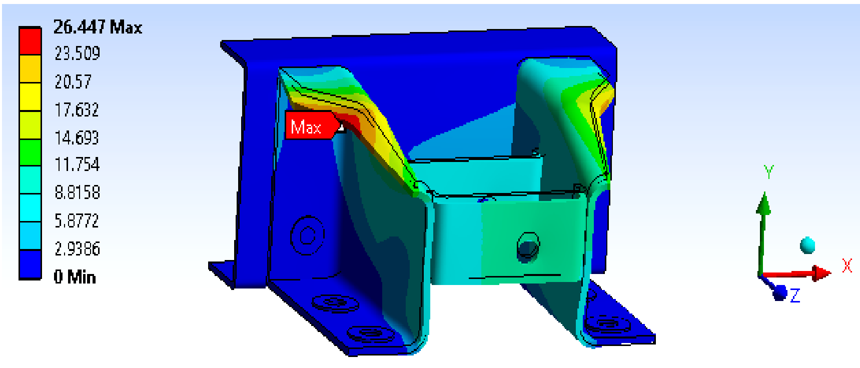
Modal analysis was performed in the ANSYS software. The same boundary condition of bracket assembly, as considered in the static analysis was considered for modal analysis. Following parameters were defined in software before modal analysis.

a) **Number of frequencies**: - For this study first eight natural frequencies are necessary. This is decided by the modal effective mass concept. This concept describes that how many modes to be extracted during modal analysis to calculate accurately the transient dynamic response of system. It also tells which particular mode will contribute maximum in load analysis in particular axis. Total effective mass contribution of 8 modes in x, y, z direction is 91.01%, 90.014%, and 86.00% respectively .This mass participation is sufficient for calculating accurately the dynamic response.
b) **Damping**: - No damping was considered for this analysis.

The results from the modal analysis indicate natural frequencies and their mode shapes. These mode shapes shows the relative deformation of component at a particular modal frequency. The first eight natural frequencies and mode shape obtained through this analysis are shown in Table 5.6

Table 5.6: First Eight Natural frequencies and their Mode Shapes of Bracket Assembly

Sr. No.	Natural Frequency (Hz)	Mode Shape
1	118.07	
2	122.4	
3	230	
4	283.4	

<table>
<tr><td>5</td><td>509.12</td><td>
</td></tr>
<tr><td>6</td><td>662.41</td><td>
</td></tr>
<tr><td>7</td><td>748</td><td>
</td></tr>
<tr><td>8</td><td>892.2</td><td>
</td></tr>
</table>

First natural frequency is called the fundamental frequency .The fundamental frequency of this bracket assembly is 118.87 Hz as shown in Table 5.6. The Results of modal analysis presents normalized deformation of the structure and do not give the absolute displacement magnitude .The vibration pattern of first and second mode shape shows the bending along x-axis and twisting along z-axis respectively.

5.7.2 Transient Analysis

Transient analysis was performed to get the response of bracket assembly due to time varying dynamic loads. The bracket is attached to truck chassis and it behaves like a cantilever beam with a tyre load at end. The chassis vibrates due to road induced time varying load, which eventually transferred to the bracket. It is like a system of cantilever beam subjected to base excitation. In this study, chassis act as base that is excited by time varying load originated from the road. The time varying load on the base is obtained experimentally as discussed in chapter 4. This acquired time varying load cannot be applied directly to base due to following reasons.

a) The sampling rate at which vibrational signal was measured is quite high: - 15000 Hz and the signal was acquired for 40sec. That means one experimental signal contains 600,000 data points Transient simulation for such huge data is computationally very expensive and almost impossible by a simple PC.
b) It requires large processing power of computers and hard disk space.
c) The computation may take great amount of time in terms of weeks.

Therefore, to make the solution process computationally efficient and effective, small sample of acceleration data is used for the dynamic analysis. The samples were selected on the basis of maximum acceleration value. From the analysis of measured acceleration signal (chapter 4), it is noted that the acceleration along the vertical axes is severe than the acceleration measured in the longitudinal direction. It may also be noted that the main objective of the present analysis is to find the maximum stresses and deformation in the bracket generated due to dynamic load. Therefore, based on the severity level, dynamic load measured along vertical direction is used in the transient analysis of the bracket. Three small samples of 3,000 data points were taken from measured vibration signal along vertical direction and analysis was carried out.

These vibrational loads in the form of acceleration are used as an input to transient analysis. The output of this analysis is time varying stresses and deformation. The three small samples were picked from rural road vertical vibration signal as given in Figure 4.12.The boundary condition for Transient analysis is given in Figure 5.10

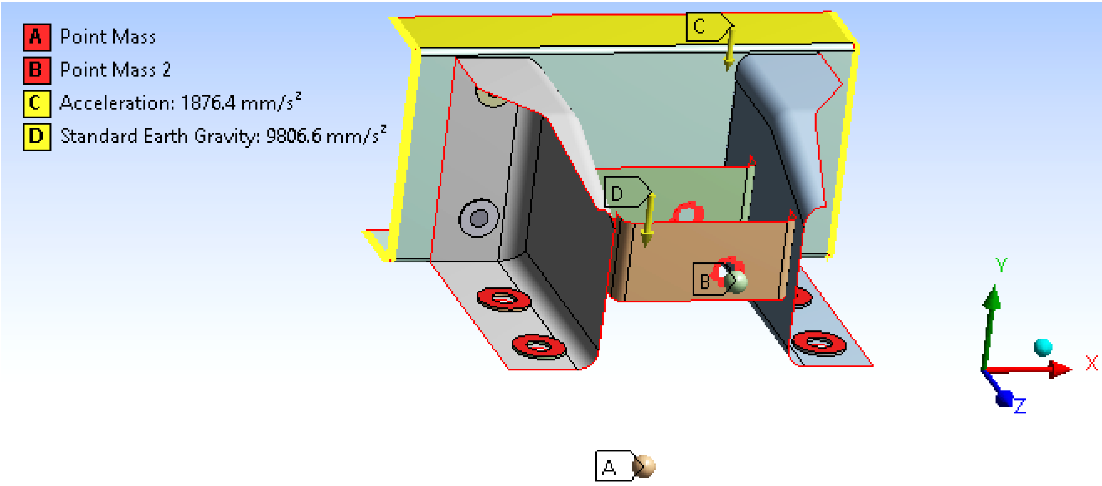


Figure 5.10: Boundary condition for transient analysis in ANSYS

Point mass A and B represent the Tyre and Shaft mass respectively. Boundary condition C is acceleration loading as sampled from vertical vibration signal for rural road. Sample 1 acceleration load is shown in Figure 5.11.The acceleration loading applied to chassis acts as base excitation.

Following conditions and parameters are required for Transient Analysis

1) **Initial condition**: It points to modal analysis, because response to acceleration load in transient analysis is calculated by using Modal superposition method and this method requires modal analysis parameter to calculate the response.
2) **Damping** :A constant damping ratio of 5% is considered for this analysis.[27]
3) **Step Time**: 6.66E-05 sec
4) **Step end Time** :- 0.2 sec

After defining the boundary condition and parameters transient analysis was performed in ANSYS. The results obtained from this analysis are Von-Mises Stress v/s time, Deformation v/s time graph shown in the Figure 5.12 and 5.13 respectively. The Maximum Von Mises and Total Deformation distribution in Bracket is shown in Figure 5.14(a) and 5.14(b) respectively.

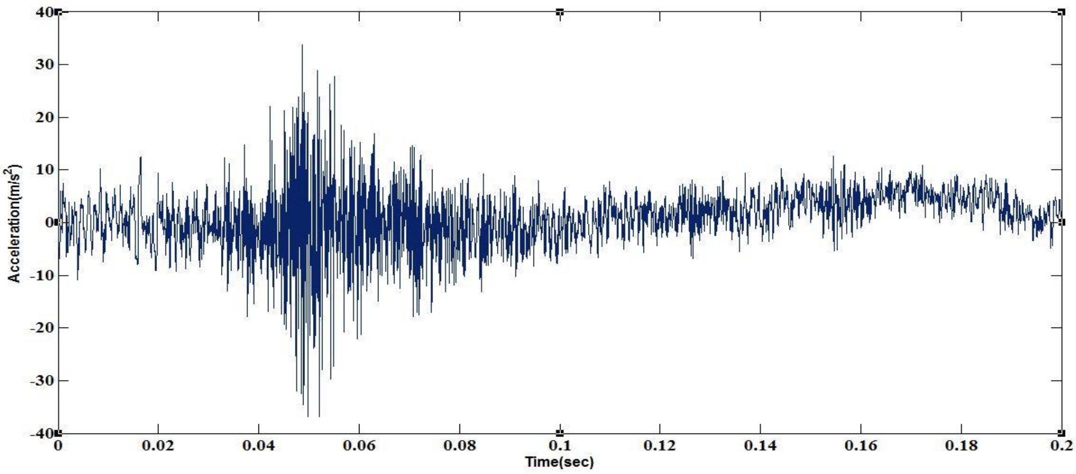


Figure 5.11: Acceleration Sample 1 for transient analysis

a) **Results of Transient Analysis for Sample 1**

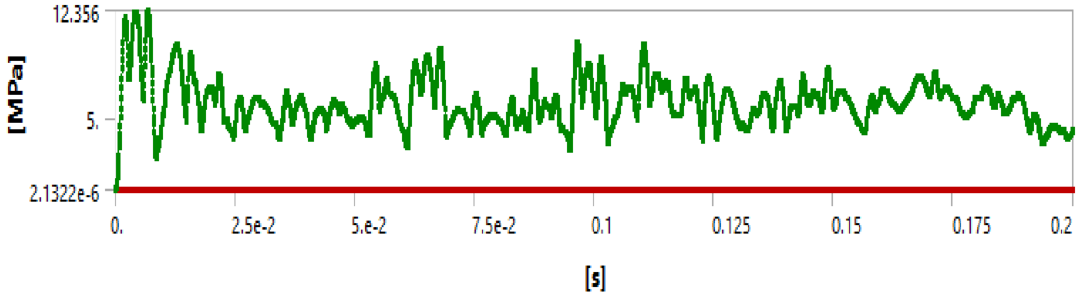


Figure 5.12: Von-Mises Stress V/s time for Bracket

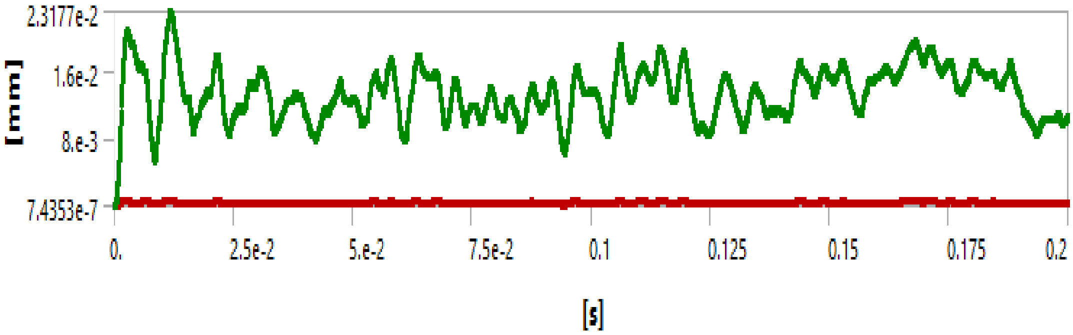
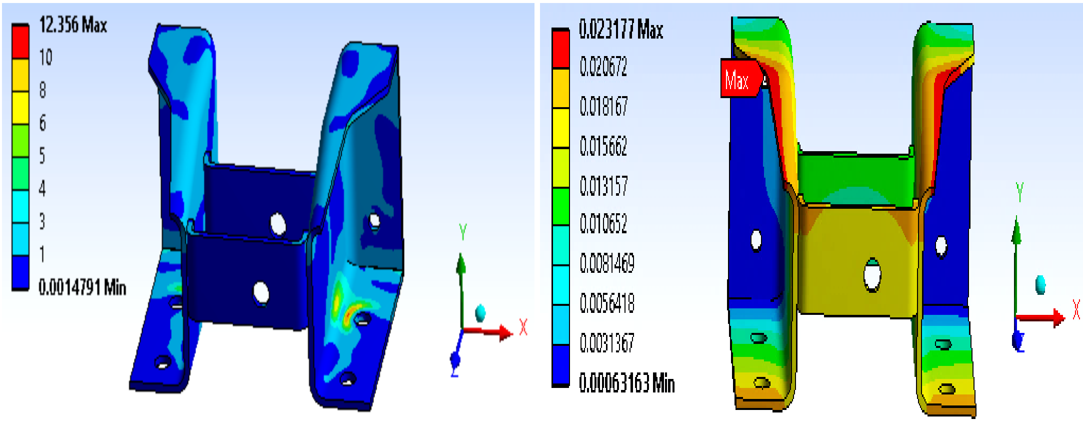


Figure 5.13: Total Deformation V/s time for Bracket



(a) (b)

Figure 5.14: (a) Maximum Von Mises Stress (in MPa) (b) Total Deformation (in mm) distribution in Bracket for Sample1

Transient analysis of bracket for 0.2 sec displays that maximum stress of 12.356 is in between the washer (M22) and bracket contact. The stress in bracket is very low. Maximum Deformation is also very less .This shows that the bracket is safe in vibration environment.

For transient analysis of bracket assembly under sample 2, the same procedure is follow as followed for sample 1. Sample 2 (Acceleration v/s Time) is shown in the Figure 5.15

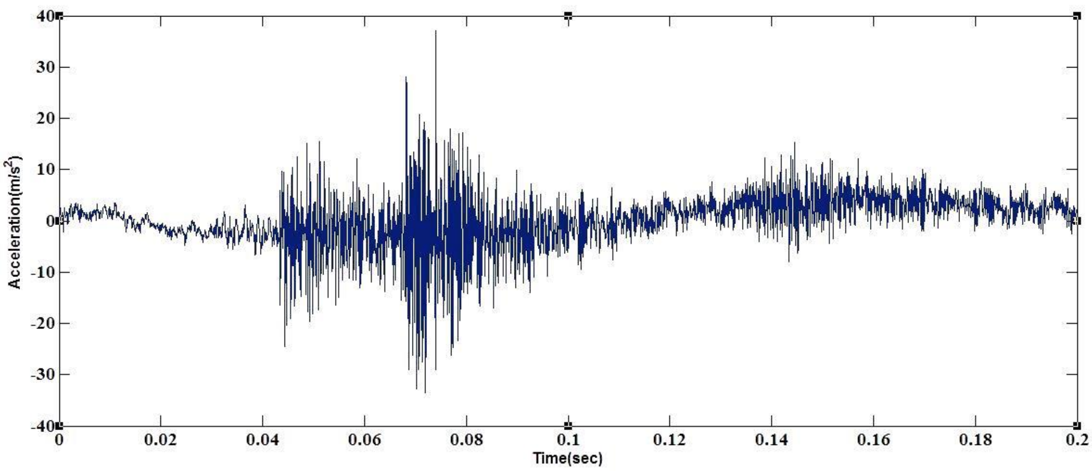


Figure 5.15: Acceleration Sample 2 for transient analysis

b) Results of Transient Analysis for Sample 2

Figure 5.16 -5.18 shows the result of Transient analysis for sample 2.

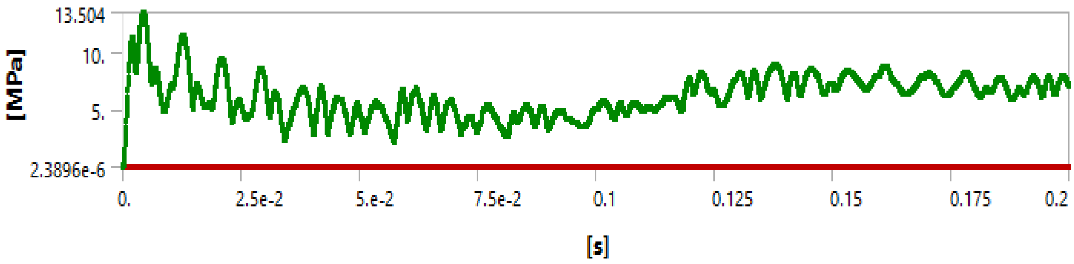


Figure 5.16: Von-Mises Stress V/s time for Bracket

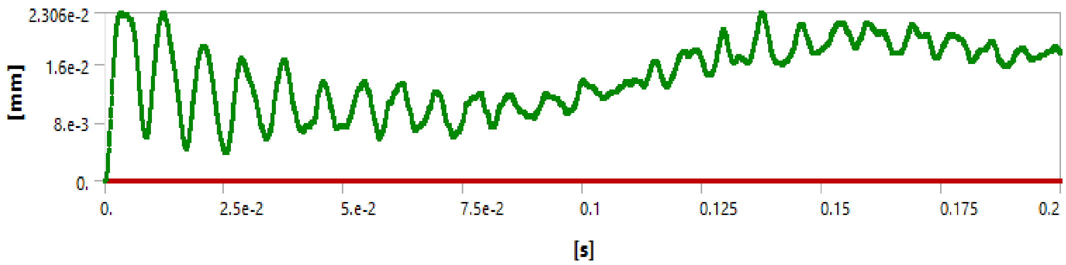
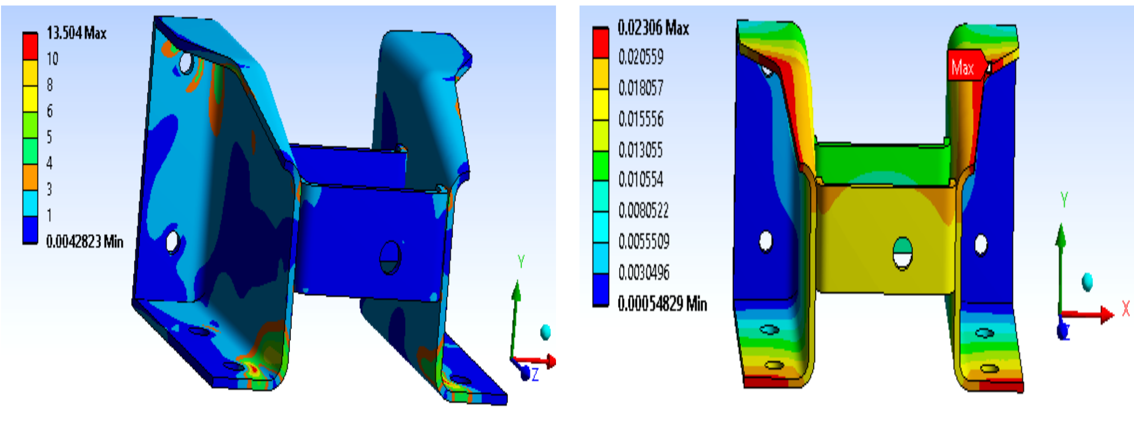


Figure 5.17: Total Deformation V/s time for Bracket



(a) (b)

Figure 5.18: **(a) Maximum Von Mises Stress (MPa) (b) Total Deformation (in mm) distribution in Bracket for Sample2**

For transient analysis of bracket assembly under sample 3 follows the same procedure as discussed for sample 1 and 2. Sample 3 (Acceleration v/s Time) is shown in the Figure 5.19

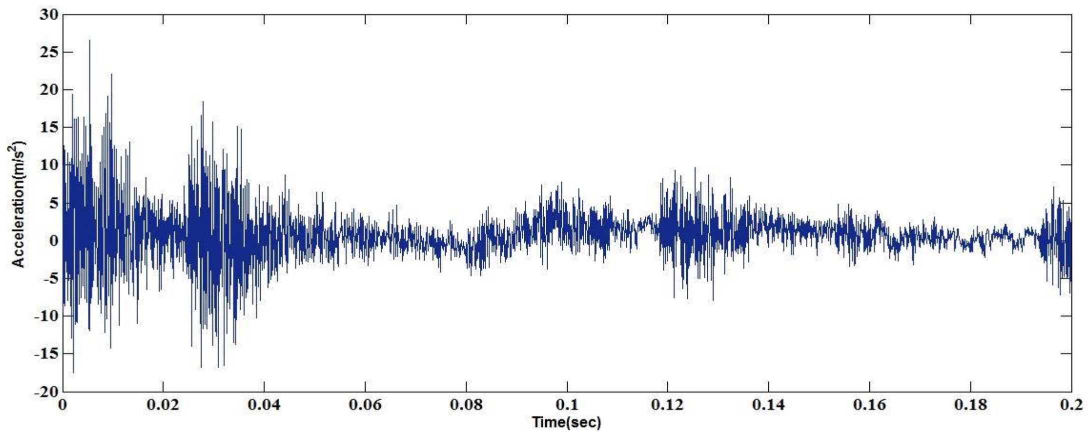


Figure 5.19: Acceleration Sample 3 for transient analysis

c) Results of Transient Analysis for Sample 3

Figure 5.20-5.22 shows the result of Transient Analysis for sample 3.

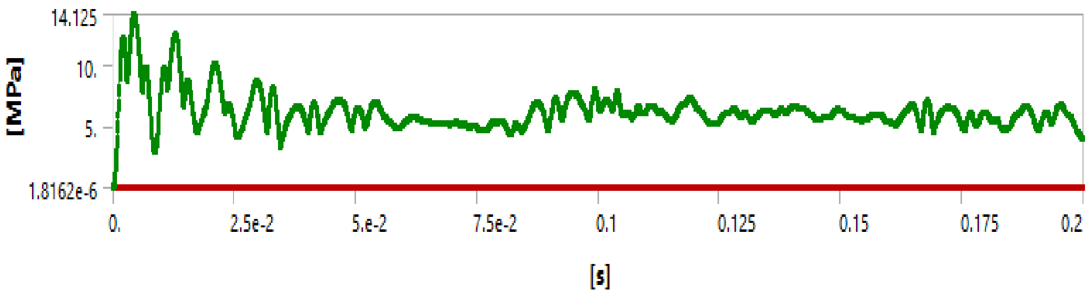


Figure 5.20: Von-Mises Stress V/s time for Bracket

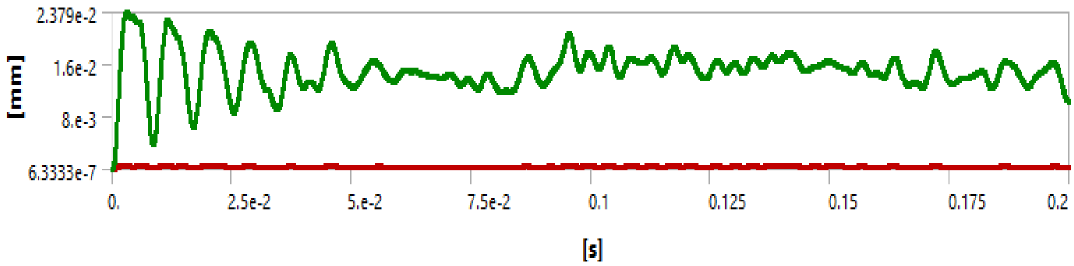
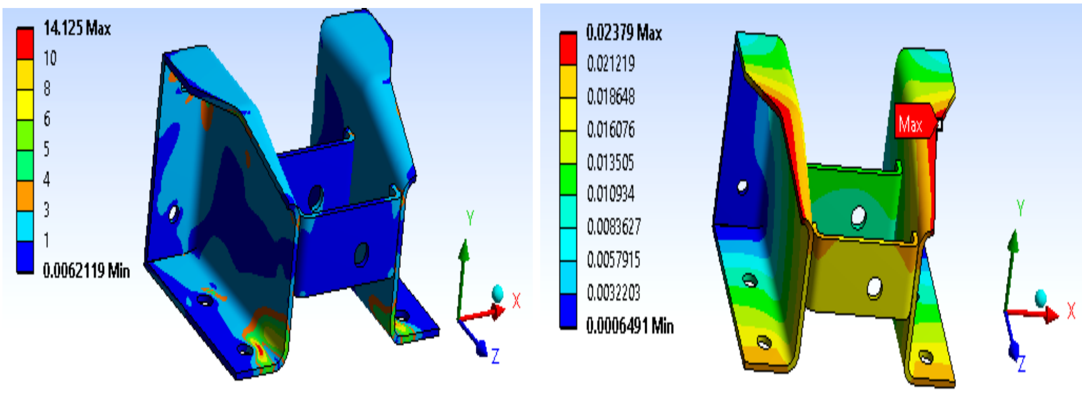


Figure 5.21: Total Deformation V/s time for Bracket



(a) (b)

Figure 5.22: (a) Maximum Von Mises Stress (MPa) (b) Total Deformation (in mm) distribution in Bracket for Sample3

Table 5.7: Comparison between results of Static and Transient Analysis

Sr. No.	Analysis Type	Sample	Maximum Stress Von-Mises(MPa)	Total Deformation(mm)
1	Static		5.1038	0.0139
2	Transient	Sample 1	12.356	0.02317
3		Sample 2	13.504	0.02306
4		Sample 3	14.125	0.02379

Table 5.7 shows that the transient analysis of bracket for three different (Acceleration V/s Time) samples, from the table, it is noted that the maximum Von-mises stress is very low than the yield stress of the material (S_{YT} = 250 MPa).Total deformation values are also very less. The difference between amplitude of the maximum stresses and deformation in the case of dynamic loading and static loading as shown in Table 5.7 is less. This shows that the structural strength of bracket in actual vibration condition. Another reason of less stress in dynamic analysis is range of excitation frequency of vibration signal that is far below from the fundamental frequency (118.87 Hz) of bracket assembly. It is observed that the transient analysis is very helpful in finding the dynamic response of the bracket in actual vibration condition. It is also concluded from Static and Dynamic analysis of bracket that the bracket is over designed and there is a scope for optimization.

Chapter 6
OPTIMIZATION OF BRACKET

6.1 Optimization of Bracket

Optimization was done with the objective of decreasing the mass of bracket. For this a parametric optimization or shape optimization was done in ANSYS software. In this a number of design parameters have to be defined manually. Design parameters can be hole size, bracket thickness, fillet radius, length of bracket etc. A single set of multiple design parameters representing a one design alternative is called design point. Like this there are number of design points. The multiple parameters are called input parameters. There are also output parameters like mass, static von-mises stress, total deformation etc. whose value is dependent upon input parameters. This optimization is done after static and modal analysis in ANSYS. The following procedure is followed to do the optimization of bracket.

1) Multiple small rectangular slots were removed from low stress regions of bracket as shown in Figure 6.1. The bracket assembly was modified in CREO software.

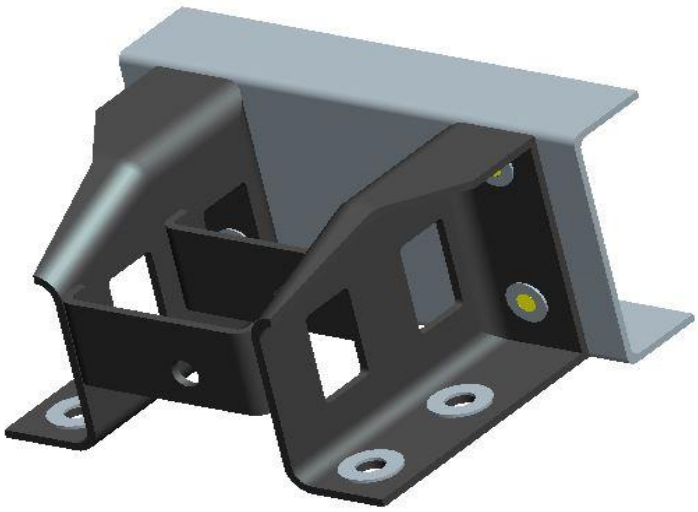


Figure 6.1: Base design for optimization of Bracket

2) By default ANSYS DesignModeler cannot change any dimension of imported CAD model. That's why any dimension cannot be used as design parameter. To change any dimension of imported CAD model in ANSYS following step are to be followed.

   a) The CAD model made in CREO software should be imported to ANSYS software by using CAD interface between them.

   b) CREO is parametric software, so every dimension is defined by unique name. To use the required dimension in ANSYS software, dimension name has to start with suffix DS .Suffix DS is also called Parameter Key in ANSYS.

   Table 6.1 shows the input parameter used in this study. These parameters are rectangular slot dimensions as shown in Figure 6.2

Table 6.1: Input Parameter Name and their base values

Input Parameter Name	Base Value(in mm)
DS_slot_height	80
DS_slot_width	50
DS_slot_dis_back	30
DS_slot_dis_base	50
DS_slot_dis	60

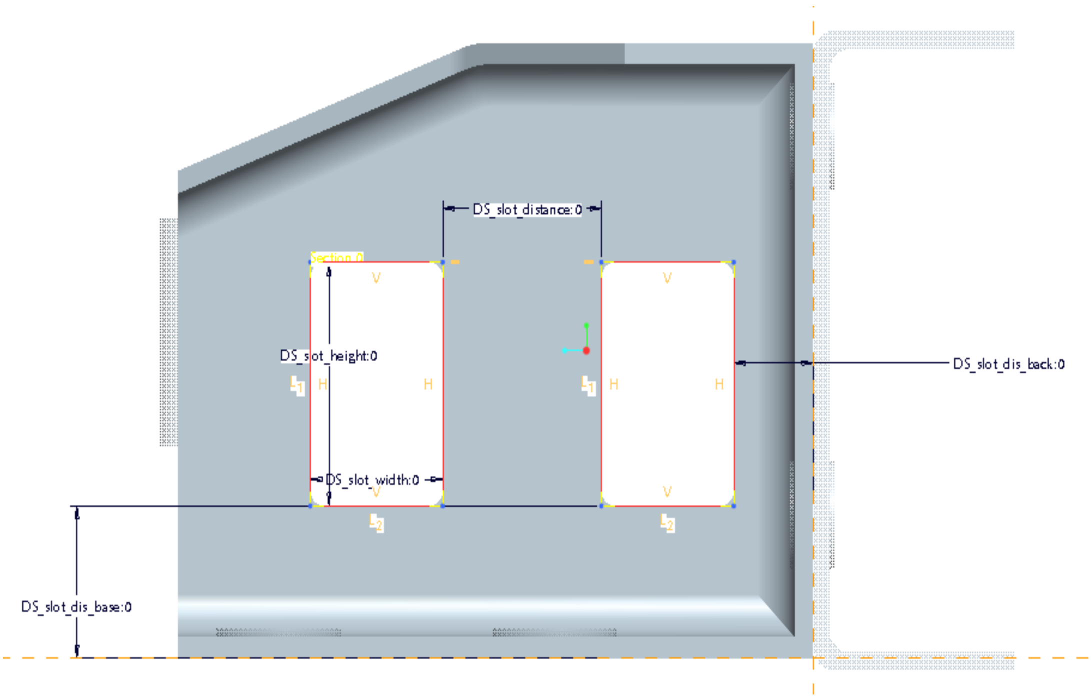


Figure 6.2: Input Parameters shown in Bracket Assembly

3) Static structural analysis and Mode analysis of base design of bracket was done in ANSYS software. The whole procedure for static and modal analysis is discussed in Chapter 5.

4) Output parameters were static von-mises stress, Total deformation, bracket mass and fundamental frequency.

5) After defining input and output parameter, a parameter interface is created in workbench project schematic as shown in Figure 6.3

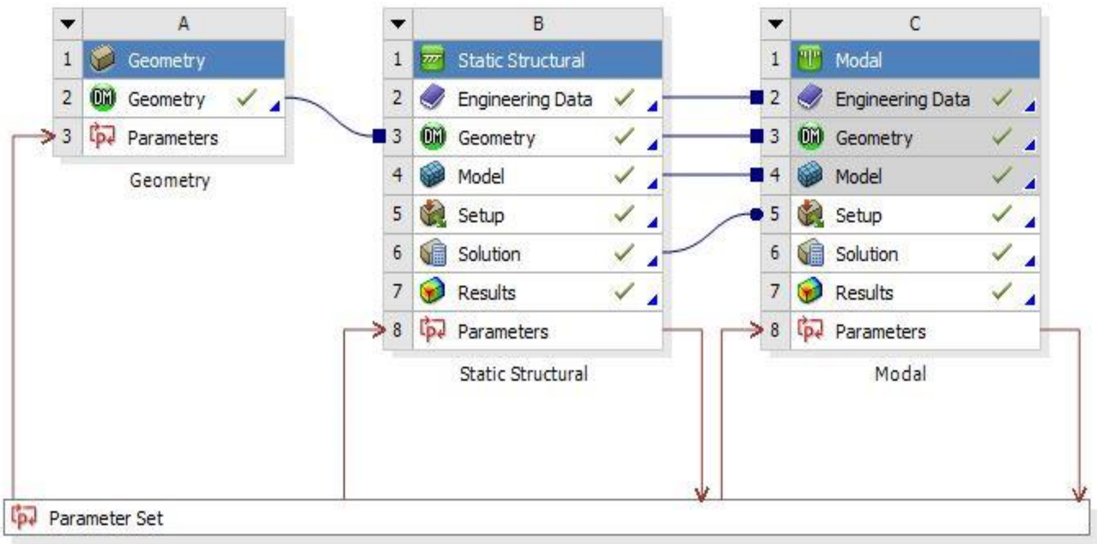


Figure 6.3: Parameter interface In ANSYS workbench

6) Values of Input parameters were added manually in parameter set window due to geometry constraints. Seven design points were used as shown in Table 6.2 .ANSYS software changes the geometry of Bracket assembly automatically according to these design points and run the static analysis and modal analysis to give the value of output parameter. These different design points were compared on the basis of output parameters. From the Table 6.2 and 6.3

Table 6.2: Design point and Input parameter values

Design point	DS_slot_height	DS_slot_width	DS_slot_dis_back	DS_slot_dis_base	DS_slot_dis
DP 0	80	50	30	50	60

DP 1	100	64	28	40	48
DP 2	100	66	27	40	47
DP 3	100	70	25	40	45
DP 4	105	70	25	35	45
DP 5	110	70	25	30	45
DP 6	110	65	27.5	30	47.5
DP 7	100	55	35	40	55

Table 6.3: Output Parameter Values

Design point	**Maximum Von-Mises Stress(MPa)**	**Total Deformation(mm)**	**Fundamental frequency(Hz)**	**Bracket Mass (kg)**
DP 0	14.7495	0.01535	112.7234	8.93
DP 1	15.3508	0.017616	102.3342	8.46
DP 2	15.4318	0.017871	101.2345	8.42
DP 3	15.6132	0.018628	98.783	8.341
DP 4	15.8107	0.019696	99.3456	8.27
DP 5	16.0590	0.021088	94.3245	8.20
DP 6	15.7719	0.019662	126.0706	8.31
DP 7	15.301	0.016285	107.74	8.62

it can be observed that the design point 5 gives the minimum value of mass of 8.2 Kg. The other output parameter was also compared with their respective limits. In this way number of design points can be used to get the optimized bracket. Due to bracket geometry constraints the Design Point 7 is chosen as the optimized design point. Figure 6.4 shows the optimized design of bracket.

The DP 7 is selected as the current design point which means this bracket current dimension is corresponds to DP 7 parameters.

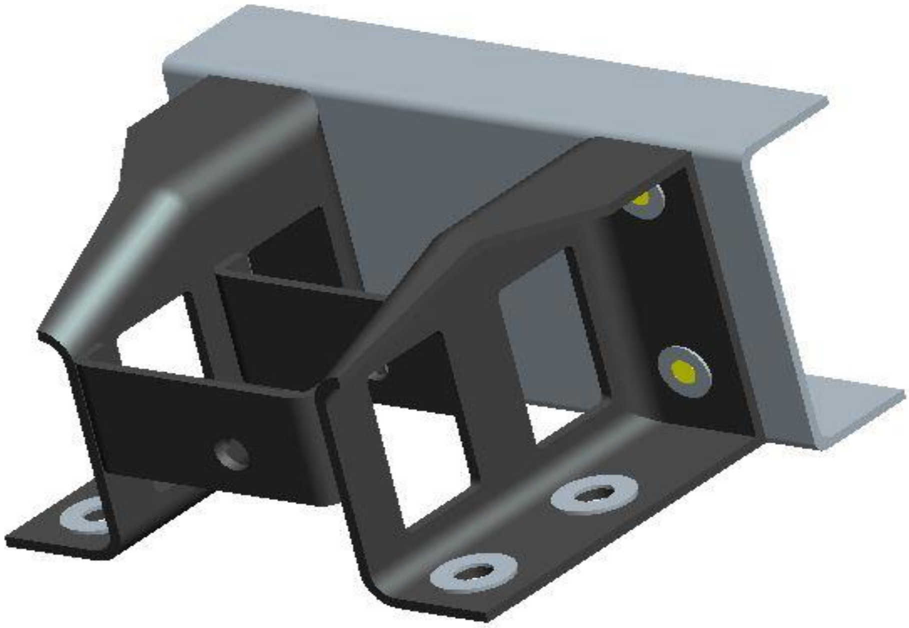


Figure 6.4: Optimized Design of Bracket

6.2 Static structural analysis of optimized bracket

The static and modal analysis was already performed for this design point by ANSYS software during optimization. The result of static analysis is shown in Figure 6.5 and Figure 6.6.

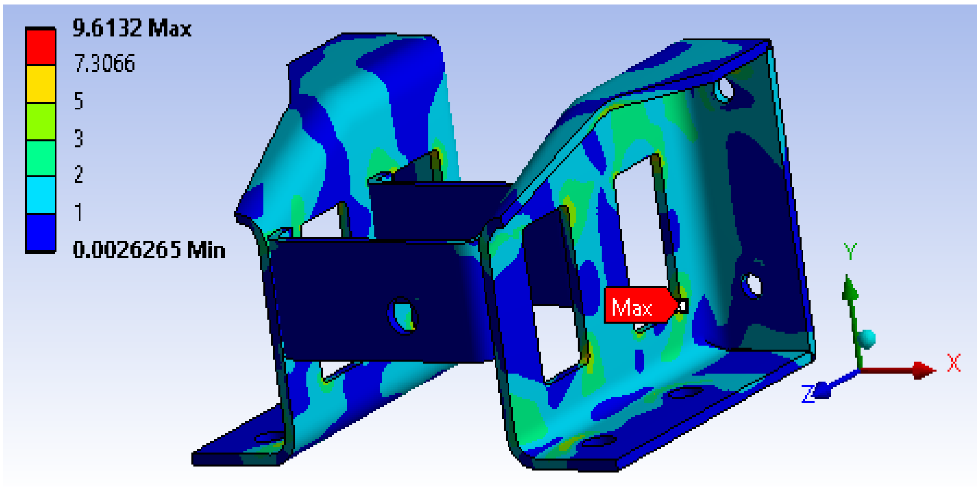


Figure 6.5: Von-Mises stress (MPa) distribution in optimized design of Bracket

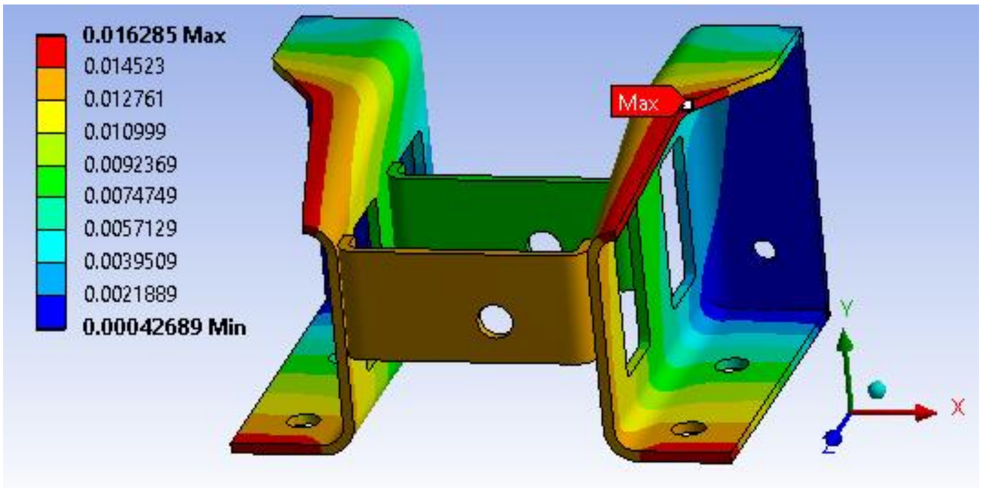


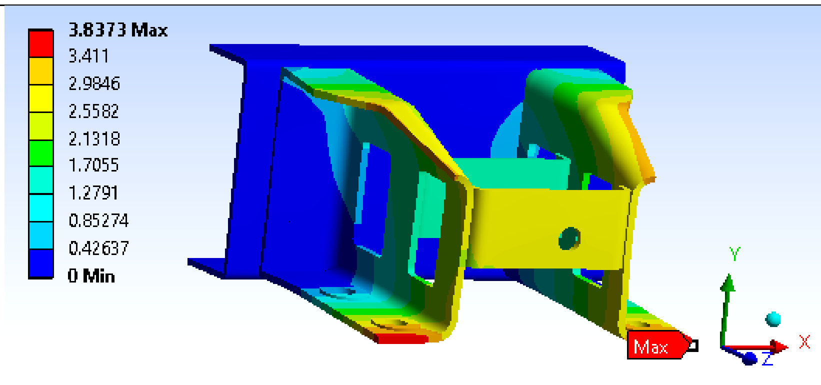
Figure 6.6: Total deformation (in mm) distribution in optimized design of Bracket

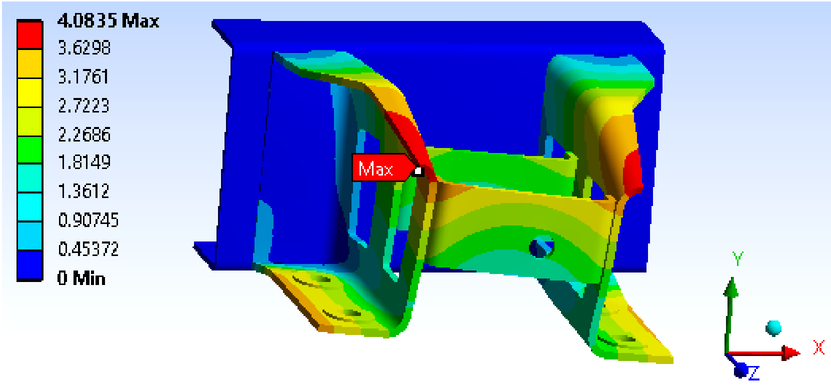
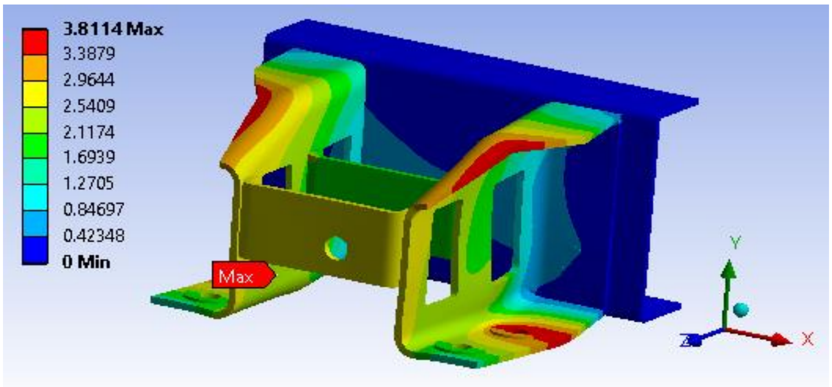
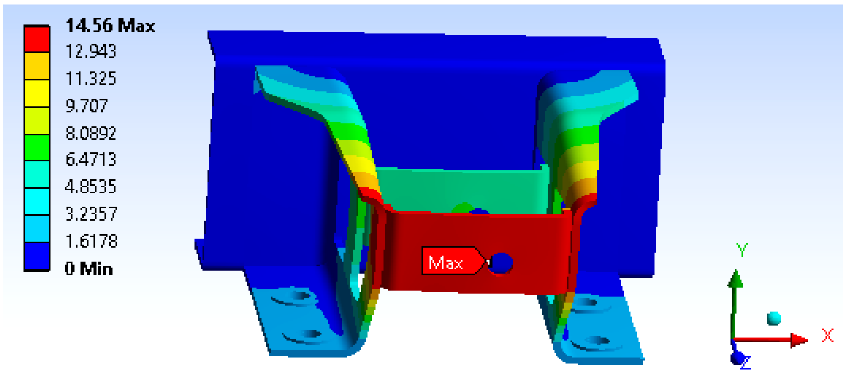
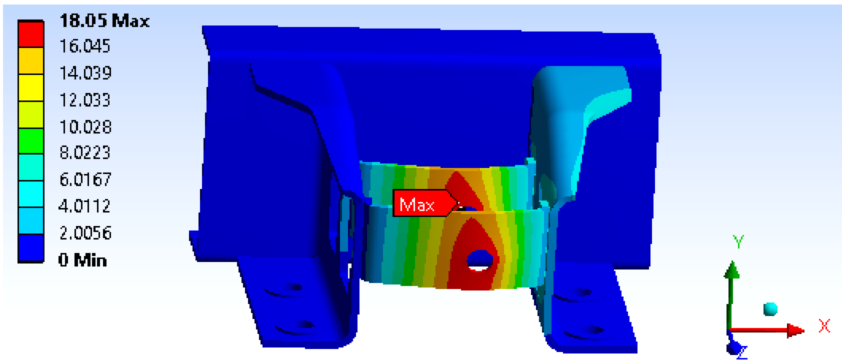
6.3 Dynamic Analysis of optimized bracket

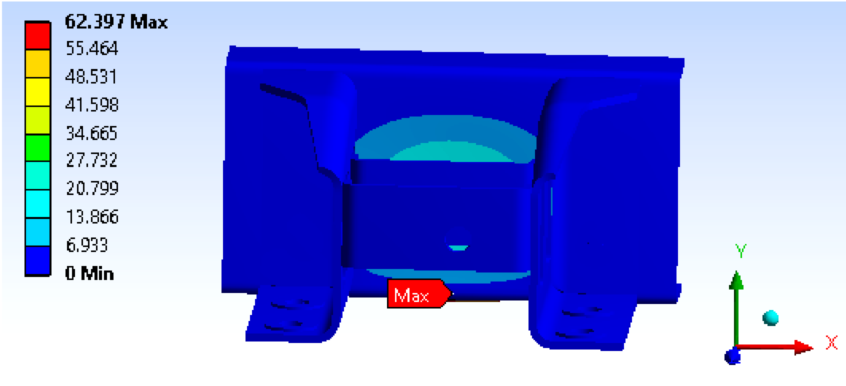
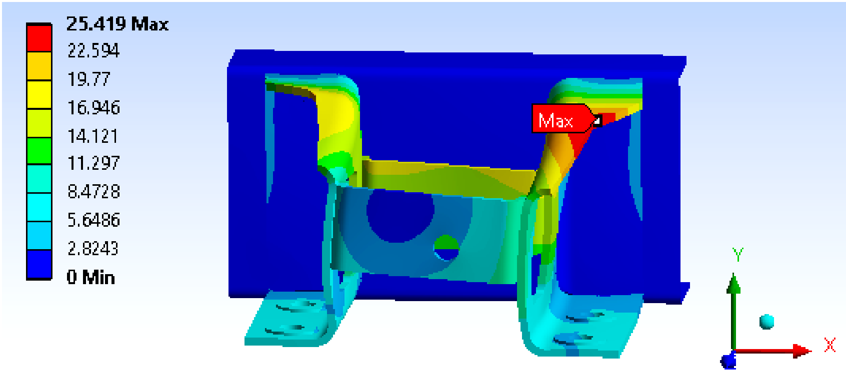
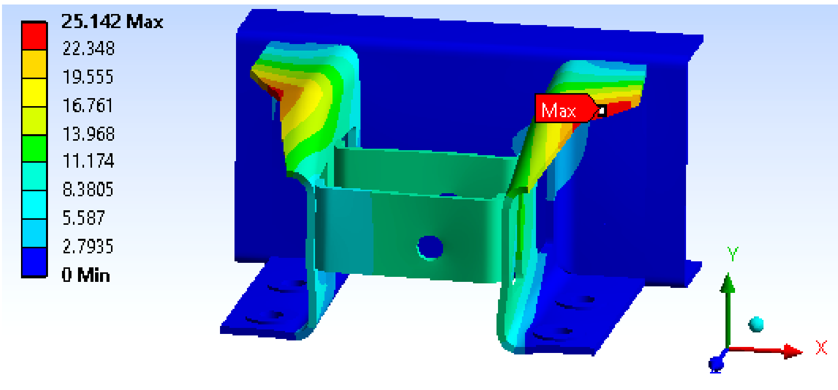
6.3.1 Modal Analysis of optimized bracket

The methodology for modal analysis for optimized bracket is same as for original design as explained in chapter 5.The first eight natural frequencies and mode shape obtained for optimized bracket is shown in Table 6.4

Table 6.4: First Eight Natural frequencies and their Mode Shapes of optimized bracket assembly

Sr. No.	Natural Frequency (Hz)	Mode Shape
1	107.74	

<table>
<tr><td>2</td><td>112.5</td><td>
</td></tr>
<tr><td>3</td><td>209.99</td><td>
</td></tr>
<tr><td>4</td><td>272.36</td><td>
</td></tr>
<tr><td>5</td><td>470.21</td><td>
</td></tr>
</table>

<table>
<tr><td>6</td><td>661.68</td><td>
</td></tr>
<tr><td>7</td><td>744.79</td><td>
</td></tr>
<tr><td>8</td><td>884.12</td><td>
</td></tr>
</table>

6.3.2 Transient Analysis of optimized bracket

Transient analysis is executed to check the validation of optimized bracket design under worst real time vibration load. In transient analysis, actual vibration environment of bracket assembly is simulated. Transient analysis of optimized bracket is performed as same as illustrated in Chapter 5 for the original design.

a) Transient analysis of optimized bracket assembly for sample 1 vibration load

Transient analysis of optimized bracket is done for same three samples as done in for original design in chapter 5. Figure 6.7 shows the sample 1 for transient analysis and Figure 6.8-6.10 shows the result of transient analysis for sample 1.

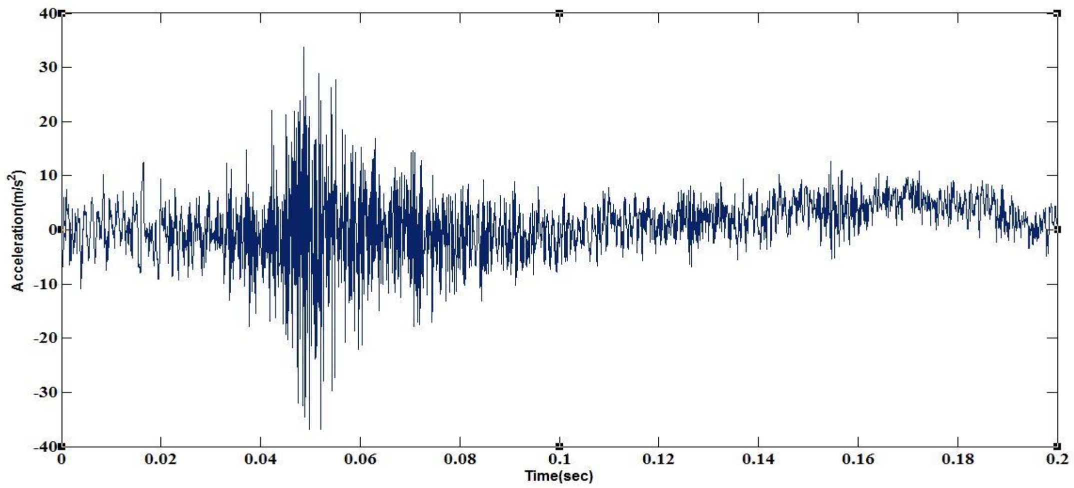


Figure 6.7: Acceleration Sample 1 for transient analysis

Results of Transient Analysis for Sample 1

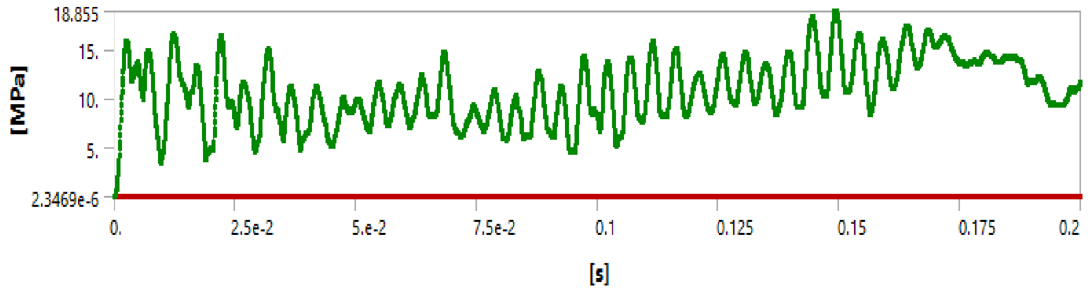


Figure 6.8: Von-Mises Stress V/s time for optimized Bracket

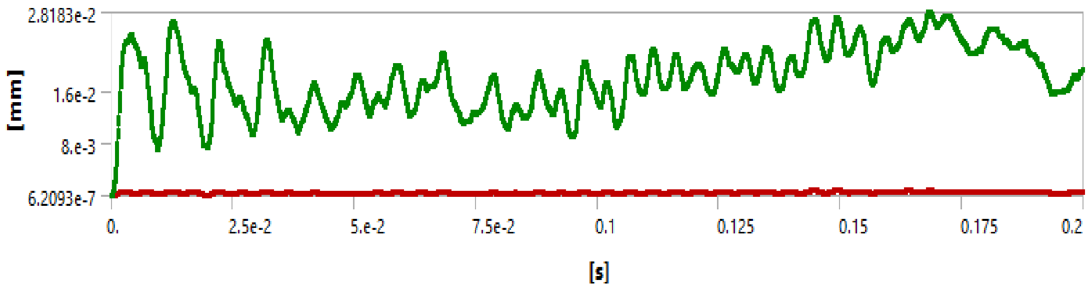
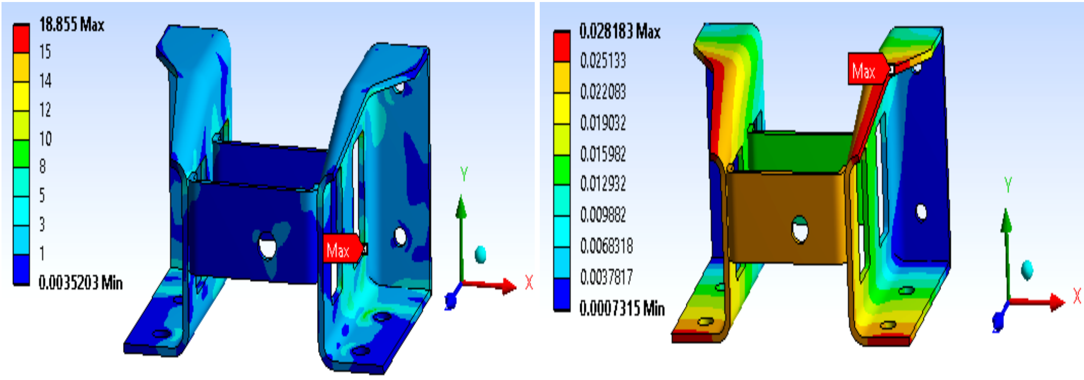


Figure 6.9: Total Deformation V/s time for optimized Bracket



(a) (b)

Figure 6.10: (a) Maximum Von Mises Stress (MPa) (b) Total Deformation (in mm) distribution in optimized Bracket for Sample1

b) Transient analysis of bracket assembly for sample 2 vibration load

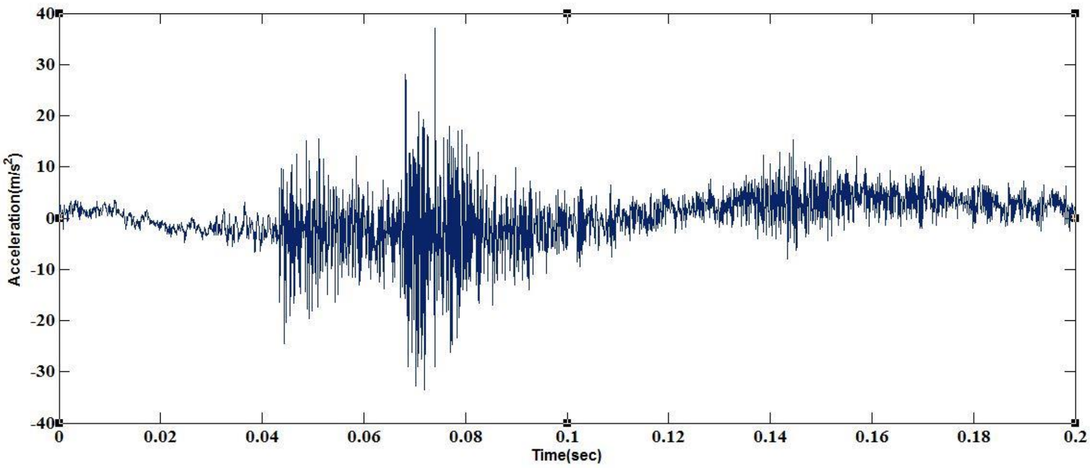


Figure 6.11: Acceleration Sample 2 for transient analysis

Transient analysis for sample 2 is performed as same as done for sample 1. Figure 6.11 shows the acceleration sample 2 and results for sample 2 is shown in Figure 6.12-6.14

Results of Transient Analysis for Sample 2

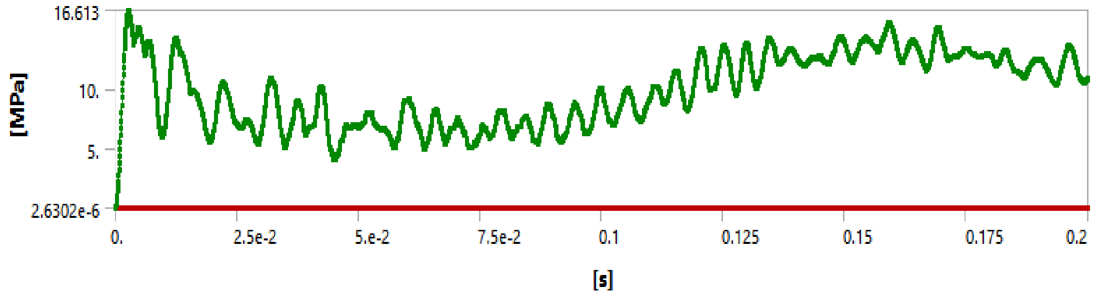


Figure 6.12: Von-Mises Stress V/s time for optimized bracket

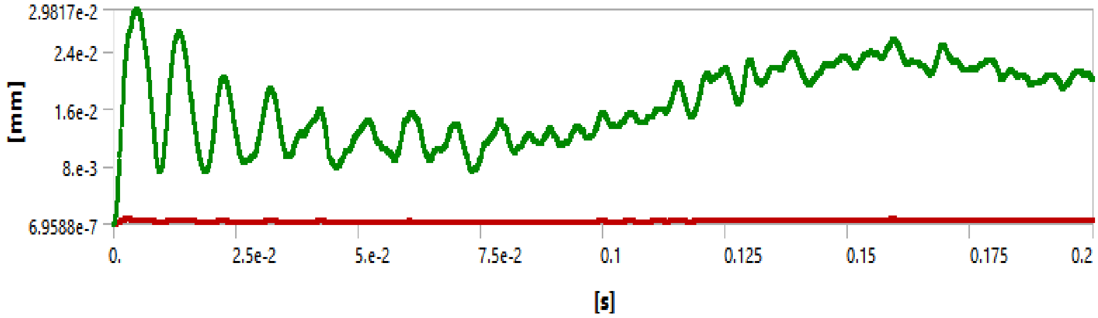
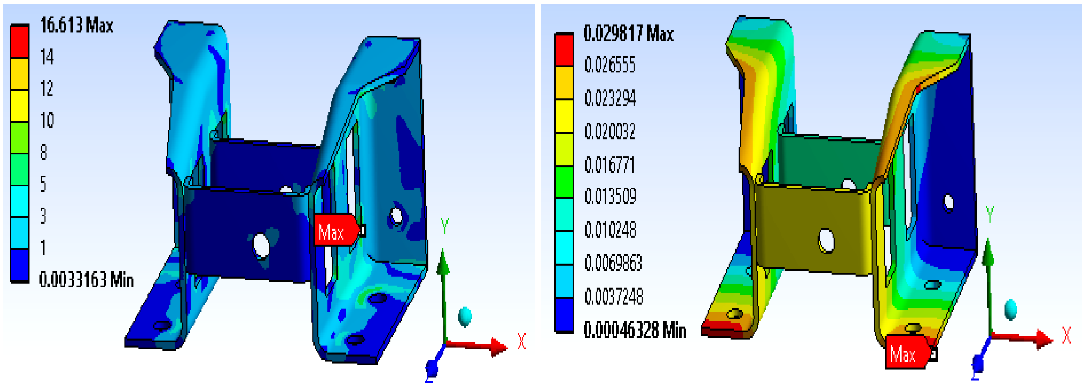


Figure 6.13: Total Deformation V/s time for optimized bracket



(a) **(b)**

Figure 6.14: (a) Maximum Von Mises Stress (MPa) (b) Total Deformation (in mm) distribution in optimized Bracket for Sample2

c) Transient analysis of bracket assembly for sample 3 vibration load

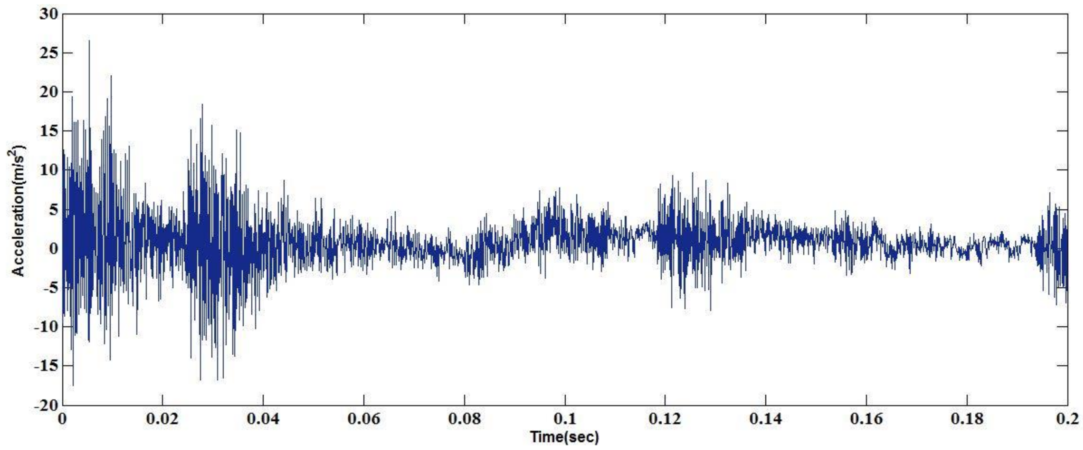


Figure 6.15: Acceleration Sample 3 for transient analysis

Transient analysis for sample 3 is performed as same as done for sample 1 and sample 2. Figure 6.15 shows the acceleration sample 3 and results for sample 3 is shown in Figure 6.16-6.18

Results of Transient Analysis for Sample 3

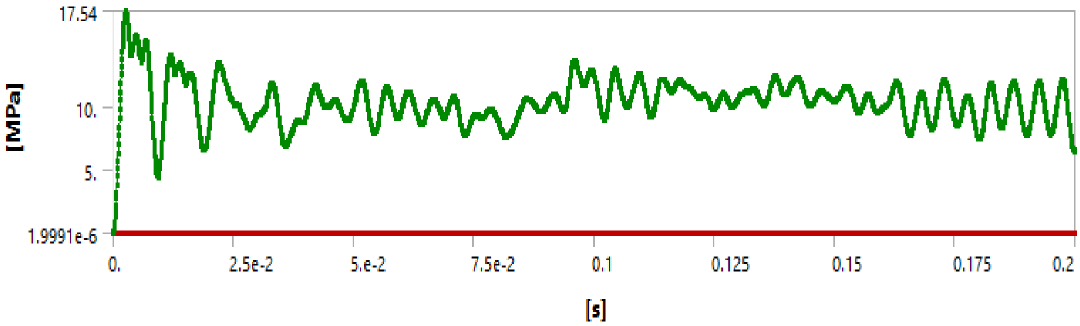


Figure 6.16: Von-Mises Stress V/s time for optimized bracket

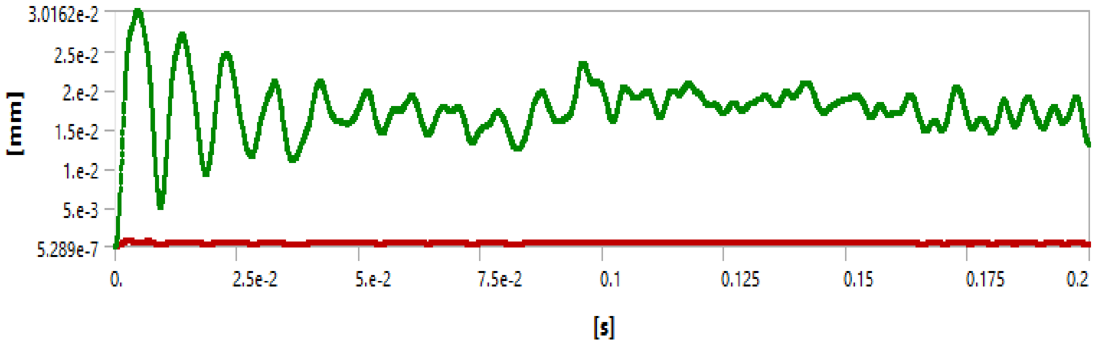
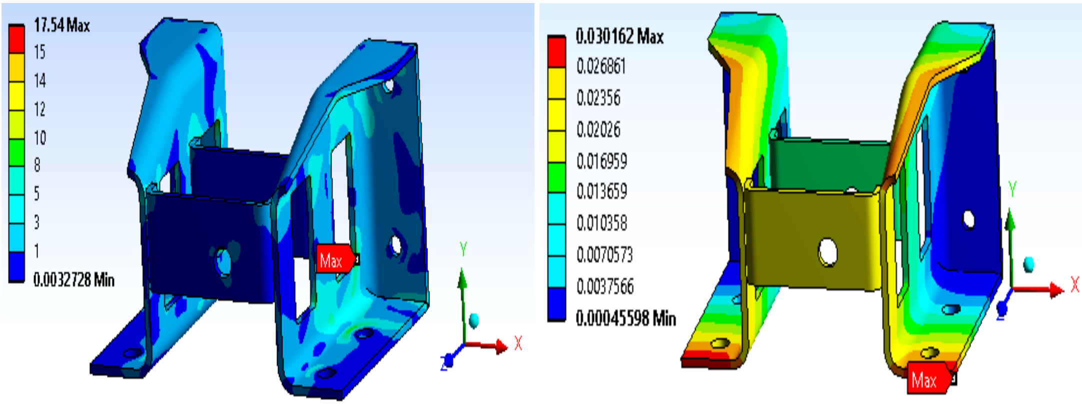


Figure 6.17: Total Deformation V/s time for optimized bracket



(a) (b)

Figure 6.18: (a) Maximum Von Mises Stress (MPa) (b) Total Deformation (in mm) distribution in optimized bracket for Sample3

Static and transient analysis of optimized bracket shows that Von-Mises stresses and total deformation is less than the prescribed limits. It shows the structural integrity of optimized bracket. A comparison between static and transient analysis result for optimized bracket is shown in Table 6.5.

Table 6.5: Comparison between static and transient analysis for Optimized bracket

Sr. No.	Analysis Type	Sample	Maximum Stress Von-Mises(MPa)	Total Deformation(mm)
1	Static		9.6132	0.01632
2	Transient	Sample 1	18.855	0.02813
3		Sample 2	16.613	0.02981
4		Sample 3	17.54	0.03016

6.4 Comparison of Results between actual design bracket and optimized bracket

Table 5.7 and Table 6.5 shows the result of static and transient analysis result for actual design of bracket and optimized bracket respectively. It can be observed from these Tables that the Von-mises stresses and total deformation for optimized bracket

Table 6.6: Comparison between mass and First Natural Frequency of Original Bracket and optimized bracket

Parameter	**Original Bracket**	**Optimized Bracket**
Mass	9.8 Kg	8.6 Kg
First Natural Frequency	118.07 Hz	107.74 Hz

increased marginally as compared to actual design of bracket. This shows the feasibility of optimized bracket design .From the Table 6.6 it is observed that a fundamental natural frequency of optimized bracket is decrease by 10 Hz as compared to original design. The maximum vibration intensity of vibration signal occurred in the range 2 Hz to 20 Hz as shown in results of chapter 4, and fundamental frequency of optimized bracket is 107.74 Hz which means there is no chance of resonance condition in bracket. This shows the safe design of optimized bracket.

Chapter 7
CONCLUSIONS

7.1 Conclusions

Optimum designing of a chassis mounting bracket in heavy automobile vehicle is an important aspect in terms of both life of the component and cost of the component. Significant research is carried out in the past to optimize a mounting bracket using static loading or dynamic loading on the bracket. Limited study is carried out on designing and optimization of a bracket using a real life road load data. In the present investigation, a problem of optimization of a chassis mounting bracket using actual road load data is performed. A numerical technique is proposed for the optimization of a bracket; where in the dynamic loading is applied from the actual measured road load data. The measured road data is of random in nature. For the present study an Eicher Turbo Truck (Model No:-11.10) under the 8 tonne category is considered.

The bracket is design for both static and dynamic loading and stresses, natural frequency, deformation etc. are analysed. Both static and transient analysis results indicate that the stresses generated in the bracket are well below its permissible strength and hence the bracket is operating within the safe limit. Both dynamic and static analysis infers that the bracket the chosen bracket is overdesign.

Therefore, optimization of bracket is carried out using commercial tool ANSYS Workbench. For the optimization, various design points are considered and for each design point, change in the output parameter (stresses) is analysed. From these various design points one design point is chosen as optimum design point. The mass of bracket was successfully reduced by 12.25%. The stresses and total deformation increased marginally in the optimized bracket. The structural integrity of optimized bracket was also checked by performing transient analysis.

Thus, it may be stated that the methodology presented in this work is general and can be used to test the structural integrity of various heavy vehicle automotive bracket subjected to different kinds of loading

7.2 Scope for the future Work

The present study can be further extended in the following ways

1) The methodology for dynamic analysis of bracket discussed in this thesis can be applied on different brackets.
2) This study can be further extended to do the vibrational fatigue analysis of bracket.
3) Different optimization techniques can be applied on the bracket.

References

[1] P.D. Jadhav, “Finite Element Analysis of Engine Mount Bracket,” *International Journal of Advancement in Engineering Technology, Management and Applied Sciences*, vol. 1, pp. 1–10, 2014.

[2] A.Janushevskis, A.Melnikovs, A. Gerina-ancane, and J. Janusevskis, “Dynamic analysis of automotive gage panel,” *International Journal of Mechanical Engineering and Automation*, vol. 1, no. 4, pp. 1–13, 2014.

[3] M. Agostinacchio, D. Ciampa, and S. Olita, “The vibrations induced by surface irregularities in road pavements - a Matlab® approach,” *European Transport Research Review*, vol. 6, no. 3, pp. 267–275, 2014.

[4] P.Wolfsteiner and W.Breuer, “Fatigue assessment of vibrating rail vehicle bogie components under non-Gaussian random excitations using power spectral densities,” *Journal of Sound and Vibration*, vol. 332, no. 22, pp. 5867–5882, 2013.

[5] S.X.Qu, D. Xu, and R. Kang, “Analysis of random vibration life of mechanical parts of actuating cylinder based on the finite element,” *QR2MSE 2013 - Proceedings of 2013 International Conference on Quality, Reliability, Risk, Maintenance, and Safety Engineering*, pp. 917–920, 2013.

[6] S. Koushik, “Static and Vibration Analysis of Engine Mounting Bracket of TMX 20-2 using OptiStruct,” in *Altair Technology Conference*, pp. 1–7, 2013.

[7] E. Mucchi, “Experimental evaluation of modal damping in automotive components with different constraint conditions,” *Meccanica*, vol. 47, no. 4, pp. 1035–1041, 2012.

[8] F. Kagnici, “Vibration Induced Fatigue Assessment in Vehicle Development Process,” *World Academy of Science, Engineering and Technology 64*, vol. 6, no. 4, pp. 88–93, 2012.

[9] S. Abdullah and N. Kadhim, “Dynamic Analysis of an Automobile Lower

Suspension Arm Using Experiment and Numerical Technique," *New Trends and Developments in Automotive System Engineering*, pp. 231 – 248, 2011.

[10] Y. Choi, D. Jung, K. Ham, and S. Bae, "A study on the accelerated vibration endurance tests for battery fixing bracket in electrically driven vehicles," *Procedia Engineering*, vol. 10, pp. 851–856, 2011.

[11] A. Khan, D. Sarkar, R. Ahmar, and H. Patel, "Random vibration analysis and fatigue life evaluation of auxiliary heater bracket," in *SIMULIA India Regional Users Meeting -11*, pp. 1–12, 2011.

[12] M. Deshmukh and K. R. Sontakke, "Analysis and Optimization of Engine Mounting Bracket," *International Journal of Scientific Engineering and Research*, vol. 3, no. 5, 2015.

[13] Z. Yanping, H. Haibo, and X. Xiaomei, "Limited Life Design of Auto Drive Axle Housing Based on CAE," *2nd International Conference on Computer Engineering and Technology*, vol. 5, pp. 260–263, 2010.

[14] O. P. Singh, T. Sreenivasulu, and M. Kannan, "The effect of rubber dampers on engine's NVH and thermal performance," *Applied Acoustics*, vol. 75, no. 1, pp. 17–26, 2014.

[15] S. K. Loh, W. M. Chin, and W. F. Faris, "Fatigue analysis of Package Terminal Air Conditioner motor bracket under dynamic loading," *Materials and Design*, vol. 30, no. 8, pp. 3206–3216, 2009.

[16] K. S. Raghavan, T. Howard, J. Buttles, B. Tool, N. Law, T. Corporation, and S. Haven, "Weight Reduction of Structural Vibration Isolation Hydro- Mount Bracket through Design Analysis and Use of Advanced High Strength Steels," *SAE Technical Paper*, pp. 1–8, 2009.

[17] W. Gang and S. Feng, "Dynamic analysis for bracket of FOG," *Proceedings - 2009 International Conference on Computer Engineering and Technology, ICCET 2009*, vol. 2, pp. 108–110, 2009.

[18] J. W. Chang and Y. S. Lee, "Topology optimization of compressor bracket," *Journal of Mechanical Science and Technology*, vol. 22, no. 9, pp. 1668–1676,

2008.

[19] S. P. Singh, A. P. S. Sandhu, J. Singh, and E. Joneson, “Measurement and analysis of truck and rail shipping environment in India,” *Packaging Technology and Science*, vol. 20, no. 6, pp. 381–392, 2007.

[20] Z. Z. Yang, “Vibration Analysis of Motor Bracket Used in Steering,” in *ABAQUS Users Conference*, pp. 1–12, 2002.

[21]..“http://www.bksv.com/Products/transducers/vibration/accelerometers/accelerometers/4533-B.” .

[22] “http://sine.ni.com/nips/cds/view/p/lang/en/nid/206677.” .

[23] F. Lu, Y. Ishikawa, H. Kitazawa, and T. Satake, “Effect of Sampling Parameters on Shock and Vibration Levels in truck transport,” *Packaging Technology and Science*, vol. 23, no. 2, pp. 101–109, 2010.

[24] R. S.Bhirud, A. K.Singh, A. V.Hotkar, and J. Rajpal, “Finite Element Analysis and Optimization of Automobile Chassis,” *International journal of Innovative Technology and Research*, vol. 3, pp. 2075–2082, 2015.

[25] P.S. Gill, *A Textbook of Machine Drawing*. S.K. Kataria & Sons, 2013.

[26] Tushar M. Patel, Dr. M. G. Bhatt, and Harshad K. Patel, “Parametric Optimization of Eicher 11.10 Chassis Frame for Weight Reduction Using FEA-DOE Hybrid Modeling,” *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*, vol. 6, no. 2, pp. 92–100, 2013.

[27] Nitin S Gokhale, Sanjay S Deshpande, S. V. B. and A. N. T., *Practical Finite Element Analysis*. Finite to Infinite, 2008.