 ISSN: 2277-9655
Impact Factor: 1.852

IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

Vibration Analysis of Loader Backhoe chassis 770 Model

Anand Thorat[*1], G.V.R.Seshagiri Rao[2], Soham Munjal[3]
[*1,2] BM College of Technology, Indore (M.P.), India
[3] Shri Aurobindo Institute of Technology, Indore(M.P.), India
anandthorat27985@gmail.com

Abstract

Construction industry is undoubtedly the backbone and propelling force behind our progress. In response to booming construction industry, utilization of earth moving equipment has increased considerably leading to high rate of failure. Backhoe Loader chassis is the skeleton of a commercial vehicles. So it is necessary to analyze chassis to avoid failure while it is in working condition. Computer simulation techniques provides a great leverage in design optimization for weight reduction, better material utilization, shorter design cycles and elimination of major part of prototype testing. The modal analysis different natural frequency of vibration and deformation pattern at each natural frequency can be found. It is necessary to find natural frequency of the structural component in order to avoid resonance phenomenon. In order to have vibration response at different location of the chassis, harmonic analysis is carried out. From this analysis the vibration pattern at different location can be observed. Hence by providing necessary change vibration level transmitted to the operator can be optimized.

Keywords: Modal analysis, harmonic analysis of Backhoe Loader chassis770 model.

Introduction

While in the running and ideal condition backhoe loader chassis subjected to various static and dynamic forces. In order to avoid resonance phenomenon it is very necessary to find out natural frequency of the chassis. Vibration causes loosening of the parts, fatigue failure of the components. Also it is very uncomfortable to operator who is working long our on Backhoe Loader. Hence to observed vibration pattern at various parts of the chassis harmonic analysis is carried out.

Objective of Present Work

The main objective of present study on "Vibration analysis of Backhoe Loader chassis 770 model" is to find out natural frequency of the chassis. Also in order to have vibration response at different location chassis harmonic analysis is carried out.

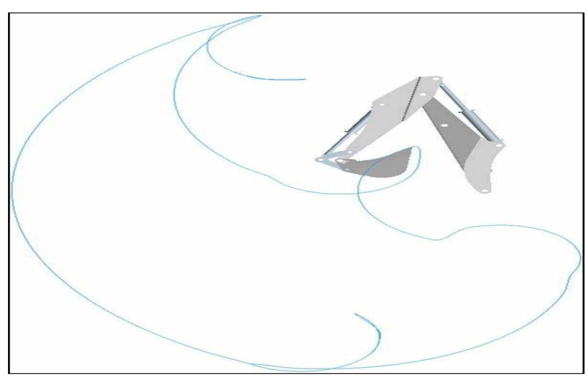
Methodology

The procedure described below has been used to obtain the objective of present study.

1) Modal analysis is carried out in Ansys wokbench
2) Harmonic analysis is carried out in Ansys workbench and in order to validate FE results compare with experimental result.

Working of Backhoe

The Backhoe is essentially, soil digging machine. The working tools off the Backhoe are actuated by the Hydraulic Cylinders. The required motion for digging operation is controlled by controlling the hydraulic cylinders. Each component is actuated by a hydraulic cylinder. A combination of extensions and retractions of the hydraulic cylinders generates the required motion of the components for digging. The hydraulic cylinder simultaneously provides the digging forces to be generated at the bucket tip. The pressure to be developed is generated by the hydraulic pump coupled to the engine. Fig. describes the working volume of such a backhoe.



Literature Review

ISSN: 2277-9655
Impact Factor: 1.852

Alan E. Duncan has reviewed vehicle modal modeling techniques is presented and applied to determine the vibration shake characteristics of a light duty pickup truck subjected to tire force variation and random road surface dynamic loads. Comparisons of natural vibration frequencies and random road response between a prototype pickup and the vehicle model are presented to establish the level of correlation. A computer optimization technique is applied to tune the body, engine, and transmission mount system of the vehicle model until the vibration level is minimized. Simultaneous reduction in road and tire induced vibration response of the modal is achieved.

Yuan Zhang et al. have studied beam elements chassis/suspension models with rigid vehicle body representation and finite element tires were under proving ground conditions. The only difference between the two models was that one used flexible beam elements and the other used rigid beams. Several proving ground road surfaces were modeled and used in the analysis, including a washboard road surface, a Belgian block type track and a pothole track,. Also analyzed were the low speed driveway-ramp and (relatively) high speed lane-change cases. The proving ground simulation results and system compliance results as well, were compared between the two models. The differences revealed the importance and necessity of using finite element model (even just using deformable beam elements) to include the component flexibility in conducting vehicle chassis/suspension dynamicanalysis.

Jeremy Lipton has observed that as vehicle development cycles become more condensed, it is necessary to perform testing as expeditiously as possible. One way to accomplish this is to perform tests previously performed over the road in a controlled laboratory environment. The Noise and Vibration engineering consulting group, a division of Roush Industries, Inc. has recently commissioned a four-wheel drive chassis dynamometer located in a hemi-anechoic test cell in order to provide manufacturers and tier suppliers a faster alternative to over the road noise and vibration vehicle level testing. As a consulting company that supports a wide variety of vehicle development needs, many unique challenges had to be overcome throughout the design and construction process. This paper identifies these challenges and presents a methodology for designing and constructing a facility to meet the broad purpose of supporting the noise and vibration testing requirements of the automotive industry.

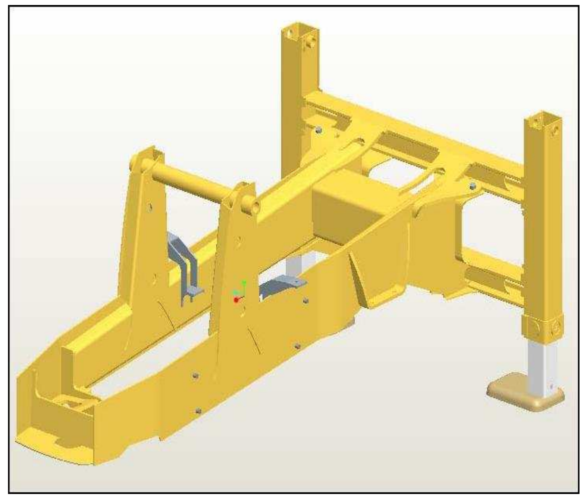
Planning of Design

Before designing any product planning of design is required and it includes following points to keep in mind.

a. Overall Size
b. Basic model characteristics
c. The way in which the model can be assembled
d. Approximate amount of components the assembly would contain e. The way in which the model can be manufactured.

Backhoe Loader Chassis

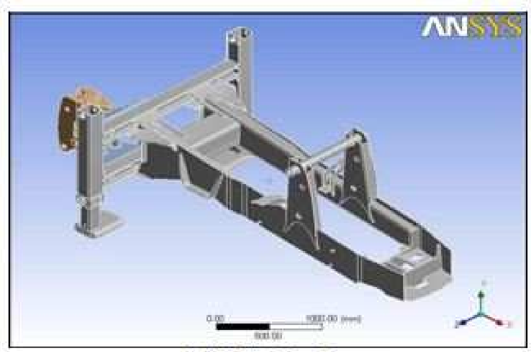
Chassis is one of the most important parts of the entire backhoe loader. It acts as a support for various accessories and mounting of engine, transmission, cab, front axle, rear axle, fuel tank, hydraulic tank etc. It consists of various plates welded together to form the entire structure.



Results and Discussion

Solid model of Backhoe Loader chassis

The chassis model is shrink wrap in the Pro/E and imported in Ansys workbench for further analysis.

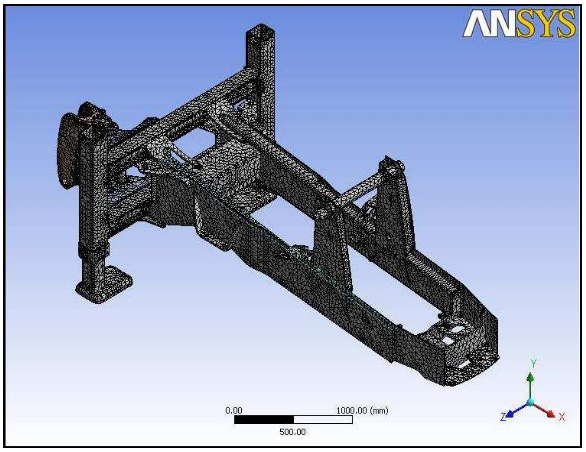


Solid Model of Boom

Finite Element model

Chassis model is meshed using tetrahedral element. There are 122583 elements and 249493 nodes.

ISSN: 2277-9655
Impact Factor: 1.852



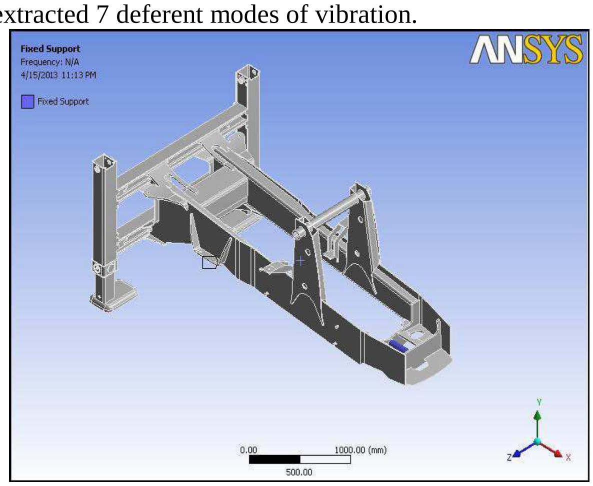
Mesh Model of Boom

Modal analysis

In order to find natural frequency and mode shape of structural component modal analysis is carried out. Modal analysis is done in free-free boundary condition. Natural frequency of the structural component should not match with the excitation frequency in order to avoid resonance phenomenon.

Boundary Condition for Modal Analysis

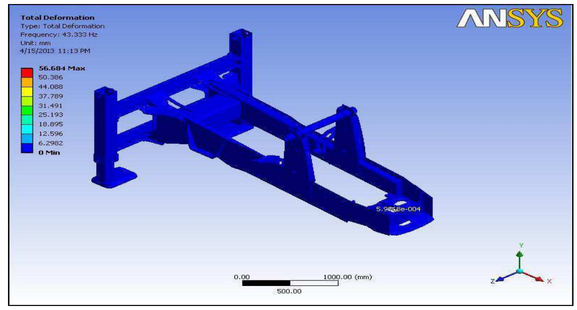
In the modal analysis chassis has fixed support at rear axle and front axle. It means that all degree of freedom at front axle and rear axle are fixed. No external force is applied to the chassis. In this analysis we have extracted 7 deferent modes of vibration.



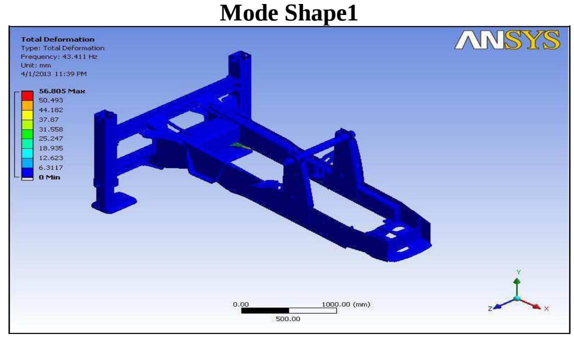
Fixed Support at front and rear axle

Different Mode Shape

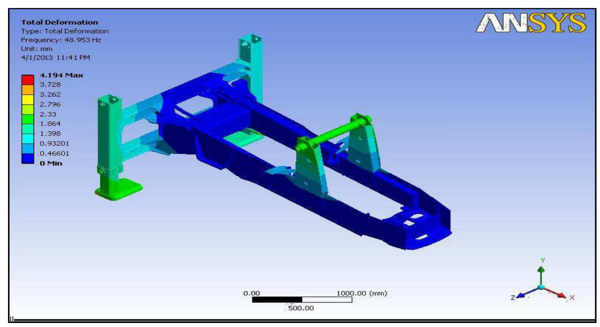
Different mode shape and deformation contour at each frequency is shown below. From the deformation contour we can observed at which frequency local vibration start in the chassis.



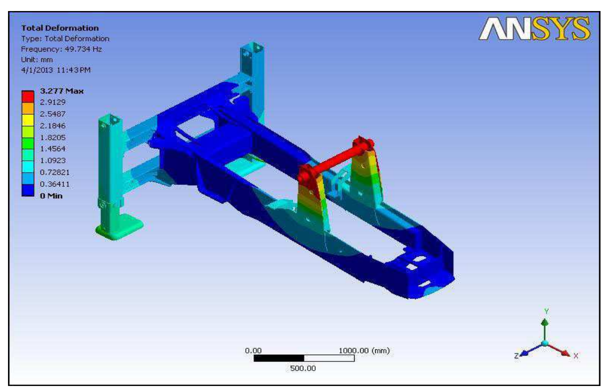
Mode Shape1



Mode Shape2

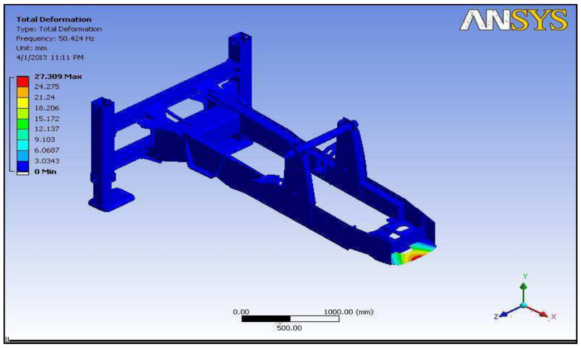


Mode Shape3

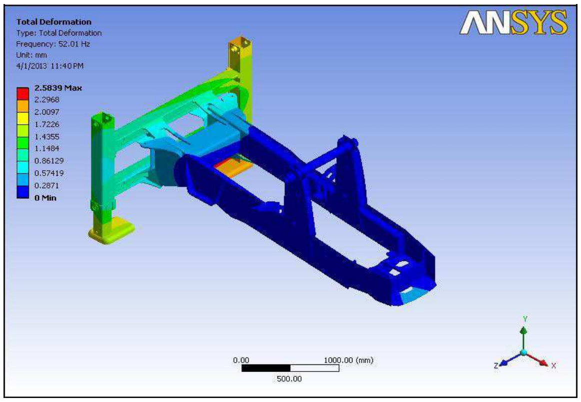


Mode Shape4

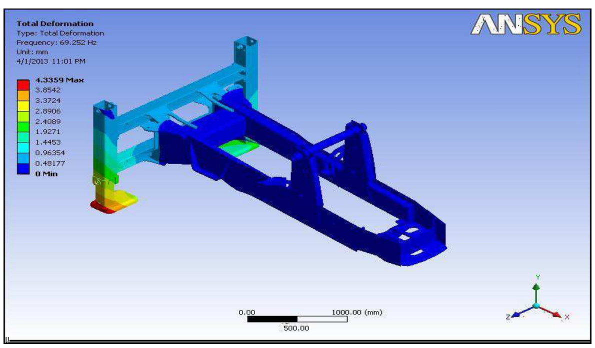
ISSN: 2277-9655
Impact Factor: 1.852



Mode Shape5



Mode Shape6

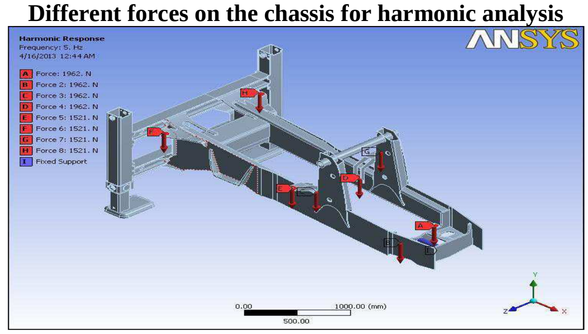


Mode Shape7

Harmonic analysis

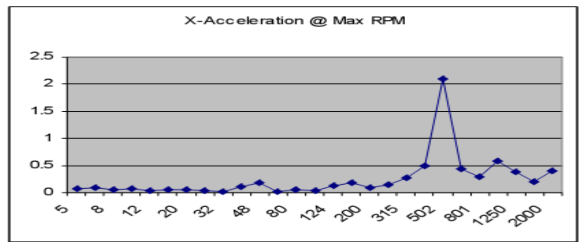
In order to get vibration response at different location of the chassis when Backhoe Loader is in ideal condition, harmonic analysis is carried out. In this case dead weight of different component is applied same as in the previous load case. This analysis is carried in the frequency range of 5 Hz to 2000Hz at solution intervals of 100Hz and vibration at different location is measured in terms of acceleration.

Different forces on the chassis for harmonic analysis

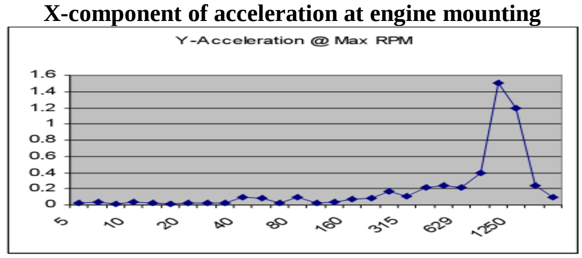


Different Forces on the Chassis for harmonic analysis

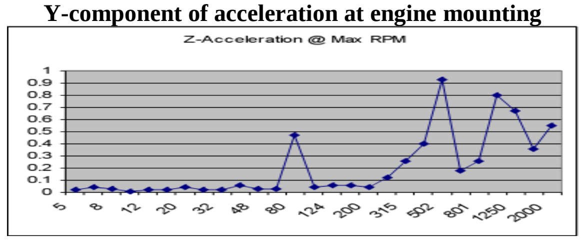
Harmonic response at different location



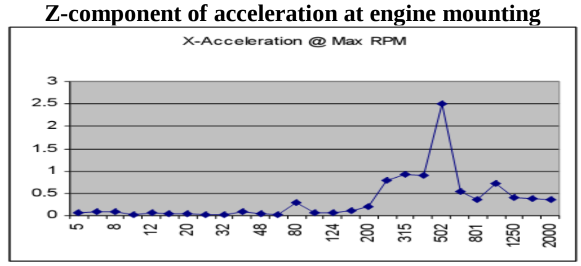
X-component of acceleration at engine mounting



Y-component of acceleration at engine mounting

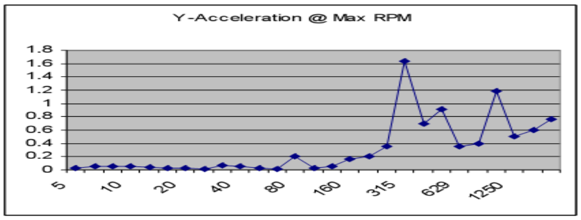


Z-component of acceleration at engine mounting

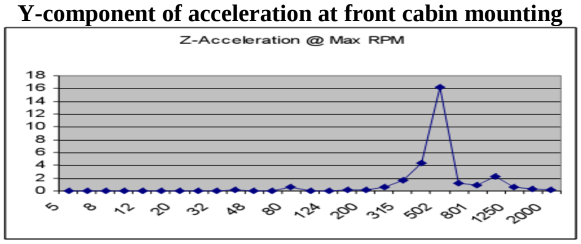


X-component of acceleration at front cabin mounting

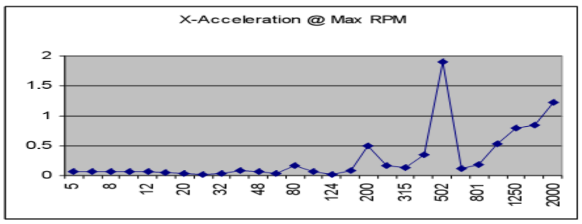
ISSN: 2277-9655
Impact Factor: 1.852



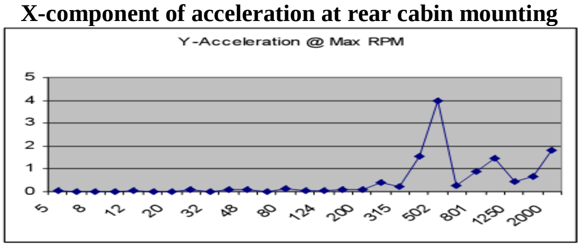
Y-component of acceleration at front cabin mounting



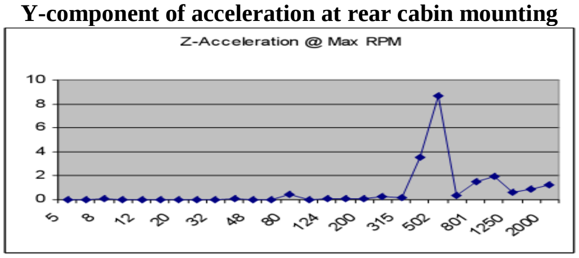
Z-component of acceleration at front cabin mounting



X-component of acceleration at rear cabin mounting



Y-component of acceleration at rear cabin mounting



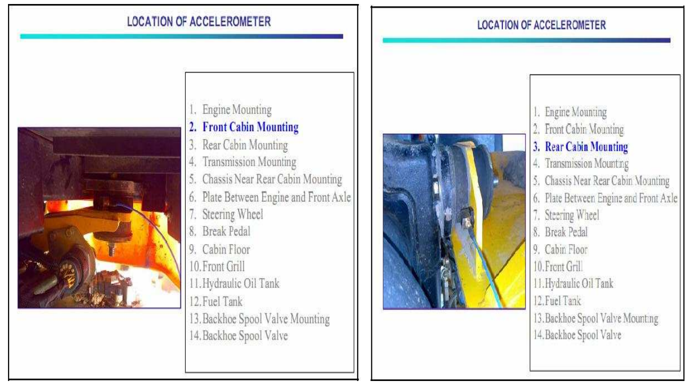
Z-component of acceleration at rear cabin mounting

From the Ansys harmonic analysis we can observe that vibration pattern in terms of acceleration at different location of the chassis. Here we can observe that maximum vibration that is all x, y and z components of the acceleration is on its peak at around 500 Hz.

Experimental vibration analysis of chassis

In order to validate harmonic analysis result of Ansys experimental vibration analysis is carried out. This analysis is carried out at 2000 constant rpm. Tri-axial accelerometer is mounted at different locations of the chassis in order to have vibration response. Accelerometer is connected to laptop via analyzer in which result of vibration in terms of acceleration is stored at particular interval.

Different Locations of Accelerometer



Accelerometer at front cabin & Accelerometer at rear cabin

From the experimental results we can observe that vibration pattern at 500Hz is on its peak value that is at engine mount, at front cabin mount and at rear cabin mount acceleration in all the x, y and z direction are maximum.

Summery

- From modal analysis we can observed that natural frequency varies from 43.33 Hz to 69.252 which is far away from the engine excitation frequency.
- In the modal analysis we can see at which frequency the local vibration starts.
- In the static analysis we can observed that due sharp edges and sharp corners stress generated at some of the location is higher than the permissible stress value. Also from this analysis we can observed the deformation pattern for each load cases.
- In Ansys harmonic analysis we can observed vibration pattern in terms of acceleration at different location of the chassis and hence conclude that all components of the acceleration are higher around 500 Hz
- In the experimental analysis we can observed the vibration patterns in terms of acceleration at around 500 Hz are higher than the other frequency same as the Ansys simulation results.

Conclusion and Future Scope

- Finite Element Analysis can be used as a tool to redesign the component if it is already designed by classical design theory.

- Without making the prototype the loading condition can be simulated and make the necessary changes at the design level, if required for the proper functioning of the component.

- Component can be optimized if it is over designed and material can be saved or some of the parts can be strengthened by adding material at proper place.
- Earth moving equipments have been known well for more then fifty years. Backhoe works well in the digging below ground level. The life of components of backhoe depends on the density of the earth, whether it is a soil digging, rock digging or it is used for mines.
- Higher stress region can be minimized by removing sharp corners, by providing smooth fillet.
- Natural frequency of the chassis is far away from the excitation frequency of the chassis.
- Ansys harmonic analysis results and experimental harmonic analysis results are quite nearer. Hence we can do the simulation without experimental setup.

Future Scope

The work done can be extended in following directions

- In the present study, the static force analysis of the backhoe loader has been carried out. It can be modified by including the inertia effect of all the components. Effect of friction can also be taken into consideration for future work.
- Fatigue analysis can be done for more accurate results and to predict the life of attachments.
- Static analysis will be carried out for other different load cases like maximum digging, loader breakout, side digging etc.
- Here harmonic analysis is carried out when the backhoe loader is in ideal condition but same analysis is carried out when the backhoe loader is in working condition.
- In future, the entire the Backhoe Loader should be taken for the analysis instead of analyzing individual component so that the cumulative effect of the entire attachment on the machine can be predicted.
- Optimization of the chassis can be done by removing unnecessary material and providing stiffener to the chassis. Also by providing proper reinforcement to the chassis vibration can be minimized so that there will be minimum vibration transmitted to the operator.

References

[1] **Alan E. Duncan** “Application of modal modeling and mount system optimization to light duty truck ride analysis” Chevrolet Motor Div., General Motors Corp.

[2] **Ann M. Nakashima** “The effect of vibration on human performance and health” Defense R&D Canada – Toronto Technical Report DRDC Toronto TR 2004-089 July 2004