Design & Analysis of Truck Engine Mount Brackets of a Truck to Improve its Performance of Life Using FEA

Deepak kumar[1], Dr. R. A. Savanur[2]

[1]M.Tech student, VTU Regional centre, Gulbarga, India

[2]Professor & HOD, BLDEA's Dr. P. G. Hallakatti college of Engg & Tech Bijapur, India

Abstract— **Engine mounting bracket assembly used in chassis front frame has been designed as a framework to support engine along with transmission member. It is a significant study which requires in-depth investigation to understand the structural characteristics and its dynamic behavior. This work presents and focuses on some Finite Element (FE) analyses performed such as normal mode analysis to determine the structural behavior. According to mode management chart the fundamental frequency is enhanced by changing the design to attain the benchmark. The most suitable model among the iterations is selected for further analyses the static analyses give the maximum structural stress condition under static loading condition. Most possible vents along Z-direction is considered. Most maximum stresses found at the bracket. So the structural fatigue analysis for Suspension mounting assembly is necessary.**

***Keywords*—suspension mounting bracket, Normal modal analysis, linear static analysis, fatigue analysis**

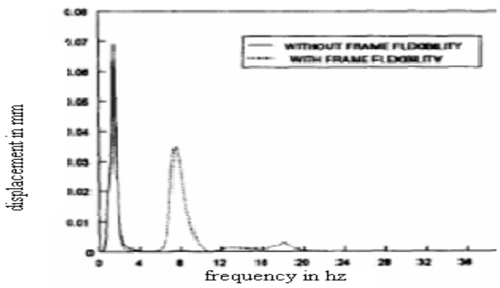
I.INTRODUCTION

Chassis is a French term and was initially used to denote the frame parts or basic Structure of the vehicle. It is the back bone of the vehicle. A vehicle without body is called Chassis. The components of the vehicle like Power plant, Transmission System, Axles, Wheels and Tires, Suspension, Controlling Systems like Braking, Steering etc., and also electrical System parts are mounted on the Chassis frame. It is the main mounting for all the components including the body. So it is also called as Carrying Unit. The chassis Frame is made up of long two members called side members Riveted/welded together with the help of number of cross members together forms an integral structure for the support of all chassis equipment and payload.Over the past years, due to global competitive demands, the automobile industries are looking for low cost and reduced weight parts with increase in performance. Hence the optimization study plays a vital role to help a manufacturer to define the mechanical parts that are lighter, reliable and lesser tendency to geometry variations. The objectives of this study are to carry out the normal mode & Static analysis of engine mounting bracket. By using the results of static analysis, fatigue analysis is carried out to predict the fatigue life of truck engine mounting bracket and weld location regions to improve the number of cycles.

II. LITERATURE SURVEY

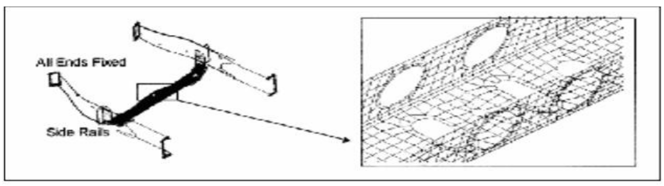
1. **Ibrahim I.M[1]**

He had conducted a detailed study on the effect of frame flexibility on the ride vibration of truck. The aim of the his study was to analyse the vehicle's dynamic responses to external factors.Hence he decided to compare between the rigid frame & flexible frame & came to conclusion as shown in the below graph



2. **Murli Krishna M.R[2]**

He has presented his study on the chassis cross member Design Using shape optimization. The problem with original chassis was that, the fundamental frequency was marginally higher than the maximum operating frequency of the transmission and drive shaft, which were mounted on these cross members. The aim of this testing was to raise the cross member frequency as high as possible (up to 190-200 Hz) so that there would be no resonance and resulting fatigue damaged. He had conducted two test; Firstly, a sizing optimization of the chassis was attempted which indicated that the mass was a predominant factor. Then four additional holes were added to the sides of the cross member to reduce its mass. Another test also have been conducted by expanding holes on the sides, the bottom holes to be reduced in size, the thickness of the attachment bracket to be increased etc. Based on all these testing, the fundamental frequency of the cross member was raised by about 4 Hz, resulting in a better design.



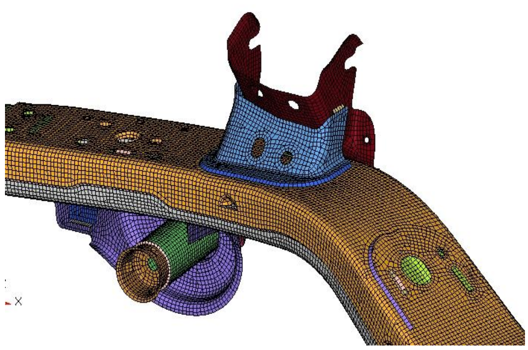
III. AIM & OBJECTIVE

1. Normal mode analysis for the engine mounting brackets assembly to attain the fundamental frequency (first natural frequency) .
2. Static analysis for the engine mounting brackets to calculate stress & displacement with suitable "g" loads.
3. Structural fatigue analysis to predict the damage on the brackets by changing the material grade to improve the life of the suspension brackets.

IV. PROJECT METHODOLOGY

1. The CAD model of engine mounting brackets assembly is provided by Ashok Leyland Limited Chennai.
2. The finite element model of the assembly to carry out finite element analysis is done using ALTAIR-HYPERWORKS v12.
3. Normal mode analysis, static analysis and structural fatigue analysis will be carried out with the application of input parameters and boundary conditions in MSC.NASTRAN-12(Normal mode analysis & Static analysis)/ N-Code(fatigue).
4. Results were obtained from the finite element method are validated by conducting experimental analysis.

V. MESHED MODEL OF ENGINE MOUNTING BRACKET



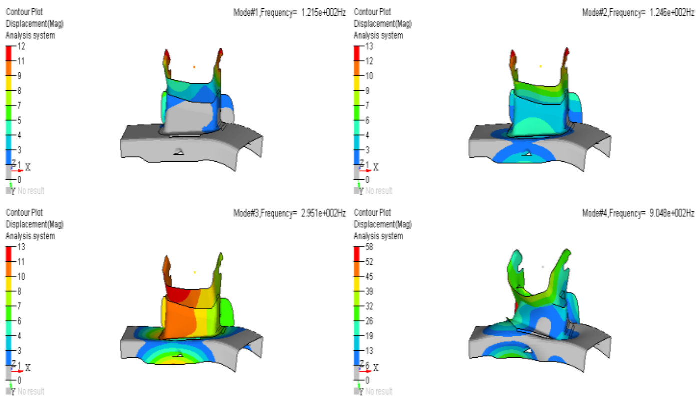
VI. RESULTS & DISCUSSION

- i. NORMAL MODAL ANALYSIS

Table.1 Mass of the engine mount brackets for different variations

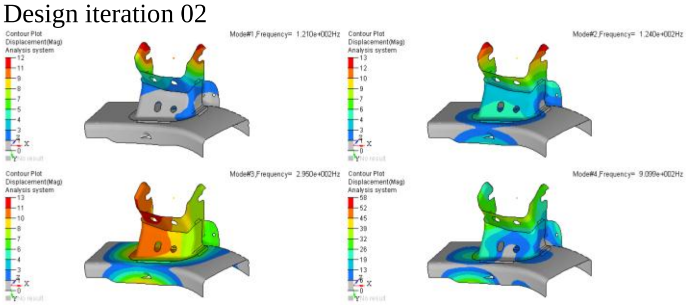
Design iteration	***Total mass of engine mounting brackets in Kg***
1	13.98
2	13.81
3	13.68

Design iteration 01



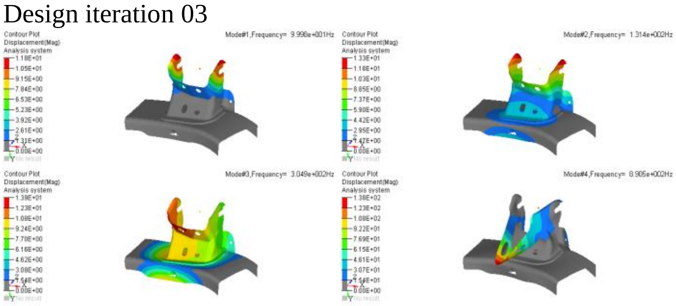
From the **Fig** shows the results obtained for the first fundamental natural frequency for **Design iteration 1** is **121.5 Hz** with total mass of suspension bracket is **13.98kg**.

Design iteration 02



From the **Fig** shows the results obtained for the first fundamental natural frequency for **Design iteration 2** is **121 Hz** with total mass of suspension bracket is **13.81 kg**.

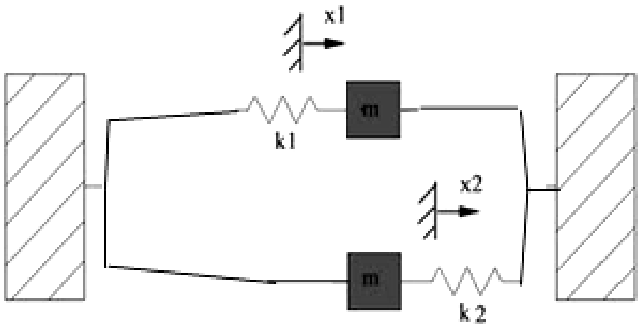
Design iteration 03



From the **Fig** shows the results obtained for the first fundamental natural frequency for **Design iteration 3** is **99.8 Hz** with total mass of suspension bracket is **13.68 kg**.

Theoretical Calculations of Modal Analysis

For analytical calculation purpose the engine mounting bracket assembly is considered into two spring and mass system which is connected in parallel.



M1 - Mass of the engine mount brackets in Ton.

M2 - Mass of the local frame in Ton.

K1 - Stiffness engine mount brackets in N/mm.

K2 - Stiffness of the local frame in N/mm.

ω_n - Natural frequency in rad/sec.

A1 - Area of the engine mount brackets in mm2

A2 - Area of the local frame in mm^2

E- Young's modulus

M1=12.22kg; M2=1.46kg

WKT

$E=2.1*10^5$

K=AE/L

For spring 1

$$K_1 = \frac{2984.493*2.1*10^5}{160}$$

$$K_1 = 39.171*10^5 \ N/mm$$

For spring 2

$$K_2 = \frac{5704.512*2.1*10^5}{380}$$

$$K_2 = 31.52*10^5 \ N/mm$$

WKT $$\omega_n = \frac{1}{2\pi}\sqrt{\frac{k}{m}}$$

….Eq-1

Where,

$$K = k1 + k2$$

$$K = 39.171*10^5 + 31.52*10^5$$

$$=70.691*10^5 \ N/mm$$

$$M = m_1 + m_2$$

$$=12.22 + 1.46$$

$$=13.68 \text{ Kg}$$

Substituting the value of M and K in equation 1

$$\omega_n = \frac{1}{2\pi}\sqrt{\frac{k}{m}}$$

$$\omega_n = \frac{1}{2\pi}\sqrt{\frac{70.691*10^5}{13.68}}$$

$$\omega_n = 114.40 \text{ Hz.}$$

Modes	FEA modes Natural frequency(Hz)	Theoretical modes Natural frequency(Hz)	% of Error
01	99.98	114.40	14.42

Comparison of theoretical and numerical analysis

From the above table it is concluding that the first fundamental frequency of design iteration 03 with mass of 13.68 kg is near to the theoretical calculation. Hence design iteration 03 is selected for linear static & fatigue analysis

ii. LINEAR STATIC ANALYSIS

Linear static analysis for the selected model (Design iteration 03) is processed using the solver MSc. Nastran and post processed using Altair Hyperworks. In linear analysis, the behavior of the structure is assumed to be completely reversible; that is, the body returns to its original position state upon the removal of applied loads. Linear Elastic Material is assumed to be homogeneous and isotropic. It is restricted to material in which stress is directly proportional to strain (linear) and to loads do not take the material beyond its permanent yield point (the material remains elastic).Here static analysis is carried out to determine the stress & displacement. Here 'g' loads are used as per the company's criteria to determine the stress & displacement & then compared with material yield strength.

Vertical (Z) = -10G (Neg)

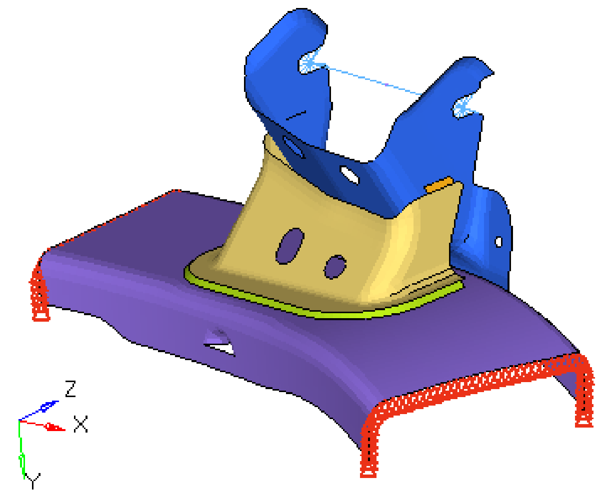
Liner(X-dir): 5G (Pos)

Lateral (Y-dir): 5G (Pos)

Vertical (Z) = -10G (Neg)

Liner(X-dir): 5G (Pos)

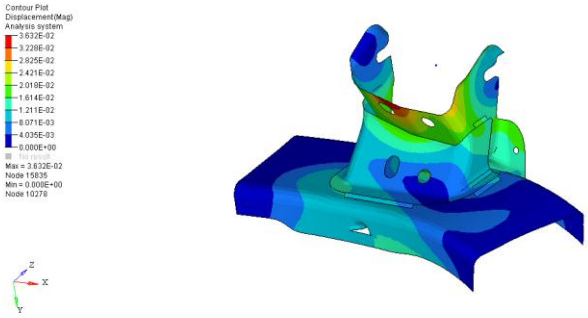
Lateral (Y-dir): 5G (Pos)



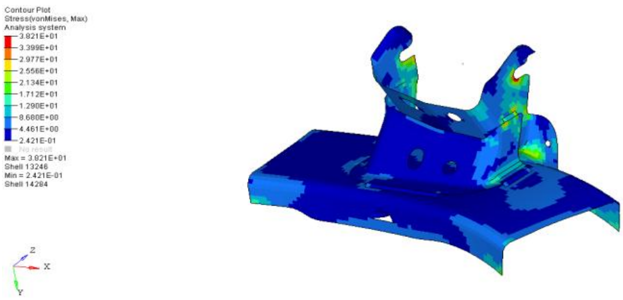
Vertical (Z) = -10G (Neg)

Liner(X-dir): 5G (Pos)

Lateral (Y-dir): 5G (Pos)



The above contour plot shows the maximum displacement for engine mounting bracket for static analysis is **0.036mm**



The above contour plot shows the maximum stress for engine mount bracket for static analysis is **38.21N/mm^2.** Whereas maximum stress of material at yield point of steel is **250 N/mm^2**

iii. FATIGUE ANALYSIS

From the above static analysis result design iteration 03 satisfies the road load criteria and the maximum displacement and stress is within the safe limit. Hence the design iteration 3 is suitable for fatigue calculation. Fatigue analysis carried out to predict the fatigue life of the component. Fatigue life estimation is carried out by the solver for two different grade materials(Body cote material & Tower material) for the design iterations 03.Here 250 cycles with 21 events are consider for fatigue calculations. The fatigue analysis is carried out using standard processor N-Code.

Body cote material properties

Material type	***Material properties***
Yield strength(MPa)	357
Ultimate tensile strength (MPa)	490
Elastic modulus (MPa)	2.06824E5
Fatigue strength coefficient (MPa)	572.1
Fatigue strength exponent	-0.057
Fatigue ductility exponent	-0.786
Fatigue ductility coefficient	2.035

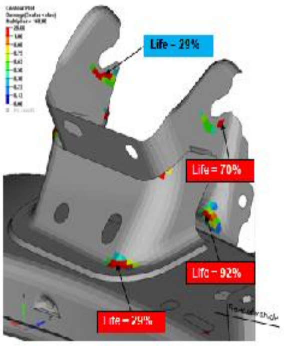
Tower material properties

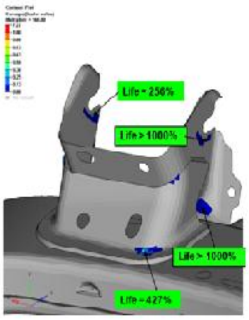
Material type	***Material properties***
Yield strength(MPa)	580
Ultimate tensile strength (MPa)	616
Elastic modulus (MPa)	2.06824E5
Fatigue strength coefficient (MPa)	1169
Fatigue strength exponent	-0.097
Fatigue ductility exponent	-1.045
Fatigue ductility coefficient	6.34

Table.2 Road load data taken for fatigue life calculation[3]

Event	Description	GVW	HPL
1	Figure 8 on asphalt	5	5
2	180 degree turnaround (4-LO / 4WD Only) on asphalt (one left + one right turn)	5	9
3	4 Corners to Call Box #1	5	2
4	Call Box #1 to Deckert	5	2
5	Deckert Rd to Soup Bowl Entrance	5	2
6	Soup Bowl w/2 passes of East Hill Road and West Slope Grades	5	2
7	Deckert to 15% Grade exit / Chalma Entrance	5	2
8	Chalma Left Lane @ 17.5 mph	5	10
9	Tank Secondary with HCS & Aggressive Launch & 100mm Bumps at 7.5 mph	5	4
10	Tank Secondary at 45mph with HCS & Aggressive Launch (AWD, 4-HI, or 4Lock)	5	5
11	Tank Secondary at 45mph (AWD, 4-HI, 4Lock)	5	5
12	Serpentine body twist	5	5
13	Chalma entrance to 32% grade entrance	5	2
14	32% Grade, up and down	5	8
15	Reverse accels to 1/4 throttle between signs at base of 32% grade (three repeats)	5	2
16	32% Grade to End of West Pritchard Rd / Beginning of South T w/ Park Brake Test	5	2
17	Shortened South Tortuous with static steers	5	6
18	South T. exit to Call Box #1 via Chalma Ext, Tank Sec, North Gravel	5	2
19	Call Box #1 to 4 Corners	5	2
20	ADR Pave w Truck Lane - 17.5 Trenches (a25 b17.5 c20 d7.5 skip e f7.5)	5	1
21	ADR Pave w Truck Lane - No Trenches, no 100mm bumps (a30 skip-b c20 d7.5 skip e f7.5)	5	1

Body cote material Tower material





comparision between the life of body cote and tower material for 250 cycles for critical parts

Body cote material in %
29
70
92
29

Tower material in %
256
>1000
>1000
427

When high cycle loads are applied on the steel component it has to sustain more than 106 number of cycles(in this project it has to sustain more than 40%). Usually most of the companies prefers this as the benchmark. Tower material sustain more cycles compared to body cote material.

VII. CONCLUSION

i. The finite element model (design iteration 03) shows an acceptable correlation with the theoretical results for measuring the natural frequencies and mode shapes.

ii. Stress & displacement obtained by linear static analysis are well below the material yield strength.

iii. For Fatigue analysis, the critical parts of Tower material grade sustain more number of cycles compared to Body cote material grade.

REFERENCES

[1]Ibrahim I.M. Crolla D.A and. Barton D.C, “Effect Of Frame Flexibility On The Ride Vibration Of Trucks”, Department of Mechanical Engineering, University of Leeds LS2 9JT, U.K. August 1994.

[2]Murali M.R. Krishna “Chassis Cross-Member Design Using Shape Optimization”, International Congress and Exposition Detroit, Michigan. February 23-26, 1998

[3] Road load data from ashokleyland.com

[4]Advanced machine design by Stephen Herman

[5]Finite element analysis by Murgendrappa.