International Journal of Advance Research in Engineering, Science & Technology(IJAREST),
ISSN(O):2393-9877, ISSN(P): 2394-2444,
Volume 2,Issue 12, December-2015, Impact Factor: 2.125

A Review Paper on Strength and Rigidity analysis of Heavy Vehicle Chassis for different frame cross section by analytically and FEA under various loading condition

Divyesh N. Chaudhari [1], Prof. Vinod M. Makwana[2], Prof. Dharmeet J. Patel[3]

[1]PG Student, Mechanical Engineering Department, GEC-Bhuj, dvsmech.5624@gmail.com
[2]Assistant professor, Mechanical Engineering Department, GEC-Bhuj, makwana_vinod@yahoo.com
[3]Assistant professor, Mechanical Engineering Department, GEC-Bhuj, dharmeetpatel@yahoo.co.in

Abstract

In this literature survey, we have found different technique to reduce the stress of heavy vehicle chassis. FEA can play important role in this analysis. The truck chassis is the main system of the vehicle and it is integrated with the main truck component systems such as the axles, suspension, power train, cab and body. The truck chassis has been offered to static, dynamic and also cyclic load. So analyze of chassis is crucial to avoid failure. 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.

Static analysis of the chassis shows the equivalent stress and deformation contour. Aim of literature review is to find out high stress area under different load condition. After that we have tried to minimize the stress by considering design aspect.

Keywords- *heavy vehicle chassis, Stress analysis, FEA.*

I. INTRODUCTION

In the era of globalization and tough competition the use of heavy vehicles is increasing for the transportation works, considerable attention has been focused on designing of the heavy vehicles. Thus it is very much necessary for the designers to provide not only equipment of maximum reliability but also of minimum weight and cost, keeping design safe under all loading conditions by careful stress analysis of the vehicles.

Heavy Commercial Vehicles or alternatively Multi Axle Vehicles having gross vehicle weight upwards of 16.2 tons. Being a life line of the economy, these vehicles are an integral part of the commercial activity of any country and these vehicles are usually deployed in the long haul distance and in transportation of materials at the ports as also in the extraction of natural resources like Iron or Coal etc.

In Modern vehicle, it is expected to fulfill the following functions:

i. Provide mounting points for the suspensions, the steering mechanism, the engine and gearbox, the final drive, the fuel tank and the seating for the occupants;
ii. Provide rigidity for accurate handling;
iii. Protect the occupants against external impact.

While fulfilling these functions, the chassis should be light enough to reduce inertia and offer satisfactory performance. It should also be tough enough to resist fatigue loads that are produced due to interaction between the driver, engine, power transmission and road conditions.

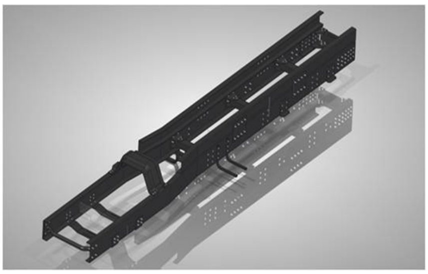

Figure 1 Heavy Vehicle Chassis

II. LITERATURE REVIEW

Cicek Karaoglu et al [1] have does stress analysis of heavy duty truck chassis with riveted joints by utilizing a commercial finite element package ANSYS version 5.3. During this study, he examine the effect of the side member thickness and connection plate thickness with length change, the side member thickness is varied from 8 to 12 mm, and the thickness of the connection plate is also varied from 8 to 12 mm by local plate, the connection plate thickness is varied from 7 to 10 mm, and the length of the connection plate (L) is varied from 390 to 430 mm. From this he concluded that if the change of the side member thickness using local plates is not possible, due to increase weight of chassis then choosing an optimum connection plate length (L) seems to be best practical solutions for decreasing the stress values.

Sairam Kothari et al [2] have studied the analysis of chassis frame for improving its payload by adding stiffener and c channel at maximum stress region of chassis frame. Analyzed was carried out for both dynamic and static load condition with the stress deflection bending moment on the Tatra chassis frame using software-Ansys. We found that with limited modifications like suitable reinforcement, increase in thickness, adding stiffeners and c channel, present payload of the Tetra can be improved from 10.4 tons to 14 tones and stress levels can be reduced from 737.3 N/mm^2 to 173.38 N/mm^2. It is less than yield stress 410 N/mm^2.

Table 1 Material property of chassis

No	Material	Yield Strength (σ_y)	Ultimate Tensile Strength (σ_u)	Young's Modulus (E)	Poisson's Ratio (ν)
1	High Strength Structural Steel	410 N/mm^2	540 N/mm^2	2,00,000 N/mm^2	0.3

Table 2 Specifications of the chassis

No	Description	Dimension (mm)
1	Length of Chassis	10208
2	Width of Chassis	1000

Roslan Abd Rahman et. al [3] does stress analysis of heavy duty truck chassis by utilizing a commercial finite element package ABAQUS. He selected ASTM low alloy steel a 710 C (Class 3) with 552MPa of yield strength and 620 MPa of tensile strength material. It was found that the maximum stress 386.9 MPa at critical point occurred at opening of chassis. This critical point is located at element 86104 and node 16045, which is in contacted with the bolt. He concluded that this critical point was an initial to probable failure.

N. K. Ingole et. al [4] did the modifications in existing model of tractor trailer. He did this by changing the dimensional parameter of cross member/longitudinal member etc.
Case-1) Variation in cross sectional areas of cross members,
Case-2) Variations in cross sectional areas of cross and longitudinal members,
Case-3) Variations in cross sectional areas of cross and longitudinal members and changing the position of cross members of main frames of chassis,
Case-4) Considering Variable cross sectional areas of cross and longitudinal members.
It has been found that, we can reduce the maximum stress form existing chassis 75 MPa to 66 MPa by case 3 and weight of chassis is reduced from 751.82 kg to 640.09 kg by case 4.

Table 3 Comparison of result for different cases

S. N.	Various Cases	Weight in Kg	Range of Equivalent Stresses on members in Mpa	Final Reduction in weight Kg	Factor of safety under sudden load (without plates)	Factor of Safety under sudden load (with plates)
1	Existing Chassis	751.82	28 to 75	-	1.66	3.37
2	Case1	705.88	17 to 69	45.88	1.78	2.71
3	Case2	674.67	22 to 75	77.15	1.66	2.71
4	Case3	663.87	25 to 66	87.95	1.89	2.31
5	Case4	640.09	42 to 75	111.73	1.66	3.37

Mohd Azizi Muhammad Nor et. al [5] performs the stress analysis on I-beam section 35 tone low loader trailer by modeling software CATIA V5R18. He selected Low Alloy Steel a 710 C (Class 3) material with 552 MPa of yield strength and 620 MPa of tensile strength. He considered uniform loaded simple beam for analysis. It was found out that there was discrepancy between the theoretical (2-D) and numerical (3-D FEA) results. It was observed that the maximum deflection point was situated in between BC1 and BC2 with magnitude of 7.79mm. The results of the numerical analysis revealed that the location maximum deflection and maximum stress agrees well with theoretical maximum location of simple beam loaded by uniform force**.**

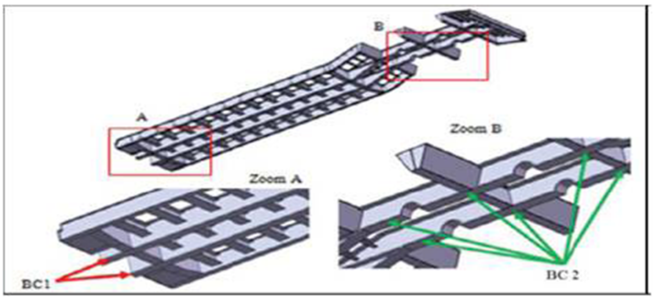

Figure 2 Boundary conditions representation on chassis of BC1 and BC2

Basia PR et. al [6] have studied the dynamic analysis of a truck chassis made of ladder type bolted cross members and side members having "c" channel reinforced with inner channel section. The chassis is given different boundary conditions with varying channel section thickness. The study is further focused on the transient analysis of the frame chassis after the preliminary modal analysis performed on the same. BSK 46 Steel material was considered.

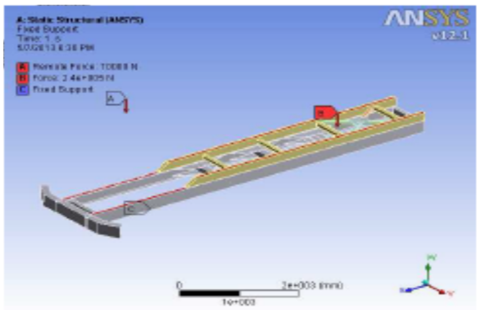

Figure 3 Static analysis

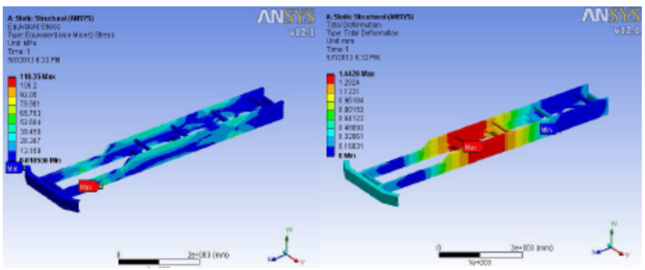

Figure 4 Equivalent stresses and total deformation

He concluded that the vibration of the chassis can be controlled through structural modification like providing the stiffeners, damping and by varying the thickness of the ladder 'c' section we can increase the load carrying capacity of the chassis.

Monika S. Agrawal et. al [7] performed optimization for weight reduction of TATA 1612 C-chassis. FEA was carried out using the ANSYS Workbench 12 for optimization under static & dynamic load. She considered Mild steel material for chassis. It was found that the mass of the chassis can be reduced from 401.55 kg to 366.76 kg (8.49%). From result, it was clear that the maximum shear stress, maximum equivalent stress and displacement are reduced and design is safe. She analyzed dynamic characteristic like the natural frequencies and the mode shapes of the truck chassis. It was found that the eight natural frequencies of the truck chassis are below 100 Hz and vary from 13 to 50 Hz. Generally most truck chassis frequencies are below 50 Hz so design is safe.

Hemant B. Patil et. al [8] Stress analysis of a ladder type low loader truck chassis structure consisting of C-beams carried by using FEM. The commercial finite element package CATIA version 5 was used for the solution of the problem.

Case 1 - Thickness of Side member is 4 mm
Case 2 - Thickness of Side member is 5 mm
Case 3 - Thickness of Side member is 6 mm
Case 4 - Change position of 4th Cross member at 2520 from rear end
Case 5 - Thickness of 5th Cross member is 5 mm.

Numerical results showed that if the thickness change is not possible, changing the position of cross member may be a good alternative. FEM results were compared with analytical calculation, where it was found that the maximum deflection by FEM was less than analytical method.

From table, it was found that it is better to change the thickness of cross member at critical stress point than changing the thickness of side member and position of chassis for reduction in stress values and deflection of chassis.

Table 4 Comparison of results

Sr. No	Analytical Method		FE Analysis		
	Displacement (mm)	Stresses (N/mm^2)	Displacement (mm)	Stresses (N/mm^2)	Weight (Kg)
1.	1.0845	123.830	0.288	71.2	141.48
2.	1.0271	100.83	0.203	84.6	165.06
3.	0.9780	85.57	0.227	91.1	188.64
4.	1.0271	100.83	0.229	77.9	165.06
5.	0.9300	100.83	0.225	40.8	167.98

Sharad D. Kachave et. al [9] have studied stress analysis and effect of web and flange's thickness on bending stiffness of a ladder type low loader truck chassis structure consisting of C-beams as a cross member. Structural analyses carried out by changing the thickness of web and flange by 1mm. The commercial finite element package ANSYS was used to analyze the stress and deflection for the solution of the problem. Web and flange thickness was optimized for bending stiffness.

The maximum deflection and stress was found to be 1.201 mm & 124.02 N/mm^2 in case 2 whereas minimum deflection and stress was found to be 0.97404 mm & 100.31 N/mm^2 in case 3. The maximum bending stiffness found to be 12011.83 N/mm in case 3 and minimum bending stiffness is found to be 9741.88 N/mm in case 2. The maximum weight found to be 557.56 kg in case 3 whereas minimum weight found to be 533.35 kg in case 4. Bending stiffness to weight ratio is found to be maximum 21.54 N/Kg.mm in Case 3 and minimum 18.13 N/Kg.mm in case 2. From bending stiffness to weight ratio it was clear that increase in flange thickness is more beneficial to increase bending stiffness as compared increase in web thickness.

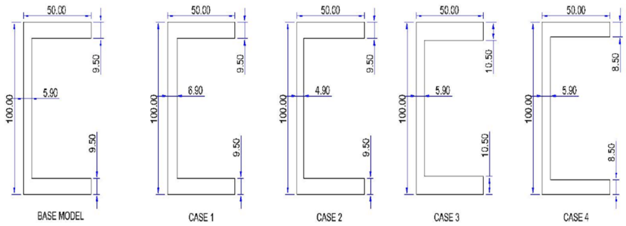

Figure 5 Dimensions of section consider for analysis

Swami K. I. et. al [10] analyzed the static structural analysis of ladder frame chassis of Eicher 20.16 using Ansys 13. He selected the chassis which has two side members or longitudinal members of 'C' cross section and seven transverse members called cross members of 'C' cross section.

He concluded that increment in load increases deformation and von misses stress linearly. As the Side member thickness increases, deformation uniformly increases. As the Side member thickness increases, Von Misses stress decrease initially then linearly increase and achieved a peak point. Thereafter it decreases linearly and again it increases linearly. The linear rates are different in increment and decrement.

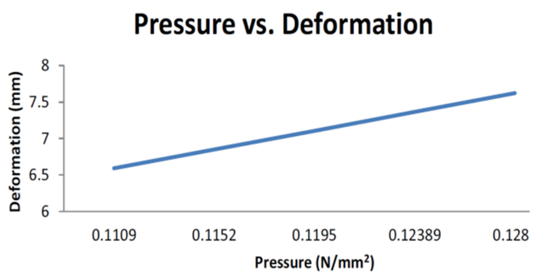

Figure 6 Pressure vs. Deformation of chassis

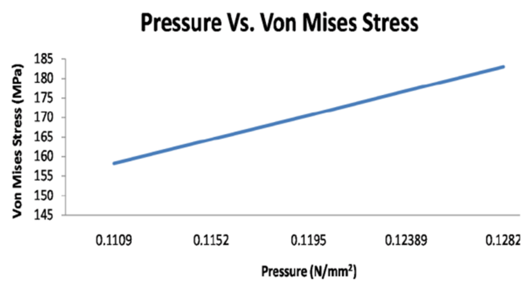

Figure 7 Pressure vs. Von Misses Stress of chassis

International Journal of Advance Research in Engineering, Science & Technology(IJAREST),
ISSN(O):2393-9877, ISSN(P): 2394-2444,
Volume 2,Issue 12, December-2015, Impact Factor: 2.125

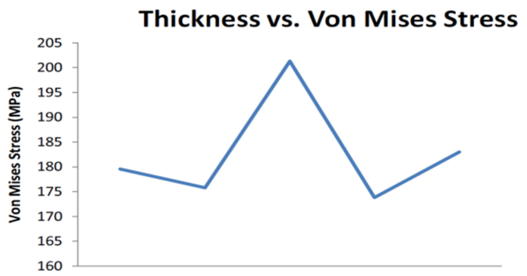

Figure 8 Thicknesses vs. Von Misses Stress

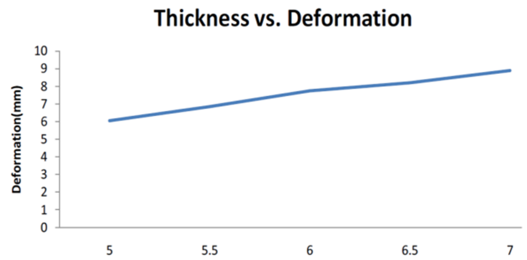

Figure 9 Thicknesses vs. Deformation

Vishal Francis et. al [11] have calculated the von miss stress and shear stress for the rectangular hollow box type chassis frame of jeep using finite element method. He considered Mild sheet steel, Aluminum alloy and Titanium alloy material for the chassis frame. The principle function of chassis is to carry the maximum load for all designed operating condition safely that should be within a limit. On chassis, frame maximum shear stress and deflection under maximum load are important criteria for design and analysis. In these projects, he concluded that

1. The generated shear stresses for all material are less than the permissible value so the design is safe for all three materials.
2. Shear stress was found maximum in mild sheet steel under given boundary conditions.
3. Von Misses stress was found minimum in Aluminium alloy and maximum in titanium alloy under given boundary conditions.
4. Mild steel material is gave good result than Aluminium and Titanium material.

Table 5 Stress analysis of ladder type chassis frame result

stresses	Mild sheet steel		Aluminum alloy		Titanium	
	Max. Mpa	Min. MPa	Max. Mpa	Min. MPa	Max. Mpa	Min. MPa
Von misses stress	29.8	0.032	28.96	0.0047	30.09	0.0075
Shear stress	16.33	0.00173	5.19	3.57	5.84	3.64
Design- stress	75.00		11.67		18.33	

Salvi Gauri Sanjay et. al [12] have performed the modal analysis and static structural analysis on the Tata 407 ladder chassis of fire truck. He selected ultra-light weight carbon fiber materials in place of traditional materials. High strength and low weight of carbon fibers makes it ideal for manufacturing automotive chassis.

From the results of steel and carbon fibers, it can be seen that von equivalent stress for carbon fibers has increased and the total deformation has reduced. Stress values for carbon fibers are under acceptable limit. Also for the same load carrying capacity, carbon fibers are preferable instead of steel for the manufacturing of ladder frame because it (mass of steel frame 170.45 Kg and carbon fibers = 54.28 kg) reduces the weight by 60-68% and increase the stiffness of the chassis frame. Hence carbon fiber can be a next option as chassis material in future as its high strength and low weight.

V. Vamsi Krishnam Raj u et. al [13] have studied the modeling and structural analysis of TATA 1109 EX2 vehicle chassis for weight reduction. The vehicle frame was initially modeled by considering 'C' cross section in SOLID WORKS 2011 and then it was imported to ANSYS 13.0. The analysis was done with three different composite materials namely Carbon/Epoxy, E-glass/Epoxy and S-glass/Epoxy subjected to the same pressure as that of a steel frame. The design constraints were stresses and deformations. The results were then compared to finalize the best among all the four frames. Present used material for chassis is steel. By employing polymeric composite materials for heavy vehicle chassis for the same load carrying capacity, there was a reduction in weight of 70% to 80%. Based on the results it was inferred that Carbon/Epoxy polymeric composite heavy vehicle chassis has superior strength, less deformation, less normal stress and less weight compared to steel, E-glass/Epoxy and S- glass /Epoxy. So he concluded that it was better to use Carbon/ Epoxy as a material for frames of heavy vehicle chassis.

Table 6 Comparison of results

Material	Mass (kg)	Max. Normal Stress (Mpa)	Max. Equivalent Stress (Mpa)	Max. Deformation (mm)
Structural Steel	385	3359	17686	5.68
Carbon/Epoxy	79	2312	16769	4.03
E-glass/Epoxy	128	2888	17055	9.45
S-glass/Epoxy	123	2888	17055	8.64

From the above literature review survey it is seen that with the help of simulation tools like finite element method, the extent of the effects of chassis input conditions can easily be estimated with minimum experimental cost.

The work involves static and dynamic analysis to determine the key characteristics of a truck chassis. The static analysis calculates the effects of steady loading conditions on a structure, ignoring inertia and damping effects. Static analysis determines the displacements, stresses, strains and forces in structures or components.

Stress can be reduced (or strength improved) by varying the thickness of web and flange in "C" section and also by varying the cross member thickness. The strength of the chassis is increased by adding the stiffener in 'C' frame section. The stress can also be reduced by changing the connection plate length and thickness.

We can use ultra-light weight carbon fiber materials in place of traditional materials in future as its high strength and low weight. By employing polymeric composite materials for heavy vehicle chassis, the weight can be reduced. The Carbon/Epoxy polymeric composite heavy vehicle chassis has superior strength, less deformation and less weight compared to other materials.

The dynamic analysis for simple structures can be carried out manually, but for complex structures like truck chassis finite element analysis can be used to calculate the mode shape and natural frequencies.

III. CONCLUSION

- Vehicle structural design and analysis has been the focus of a number of previous works. The review of some of the previously conducted work related to vehicle structural design, analysis and optimization using Ansys software is surveyed. It is found that the chassis analysis mainly consists of stress analysis to predict the weak points and fatigue analysis to predict the life of the chassis. This study makes a case for further investigation on the design of chassis using FEA Ansys software.
- Stress can be reduced by varying the thickness of side members and cross members. The strength can be improved in 'C' frame section, by changing the thickness of web and flange. Stress can also be reduced by varying the length and thickness of connection plate.
- The chassis strength can be improved by providing the stiffeners in 'C' channel section.
- From linear static analysis, maximum deformation of the component and maximum stress can be known and from that the material can be changed if required to meet the loading condition.
- The dynamic analysis can be done for finding the natural frequency and mode shape of chassis frame.
- Finite Element Analysis can be used as a tool to redesign the component if it is already designed by classical design theory.
- Feasibility of different design change & material change on stress & weight reduction can be done by FEA.

REFERENCES

[1] Cicek Karaoglu, N. Sefa Kuralay "Stress analysis of a truck chassis with riveted joints" Elsevier Science B.V Finite Elements in Analysis and Design 38 pp. 1115-1130, 2002.

[2] Sairam Kotari, V. Gopinath "Static and Dynamic Analysis on Tetra Chassis" International Journal of Modern Engineering Research (IJMER), ISSN: 2249-6645, Vol-2, Issue-1, 2002.

[3] Roslan Abd. Rahman, Nasir Tamin, Ojo Kurdi "Stress analysis of heavy duty truck chassis as a preliminary data for its fatigue life prediction using FEM" Jurnal Mekanikal, No. 26, 76-85 December 2008.

[4] N. K. Ingole, D. V. Bhope **"**Stress Analysis of Tractor Trailer chassis for Self Weight Reduction" International Journal of Engineering Science and Technology (IJEST), ISSN : 0975-5462, Vol-3, No-9, September 2011.

[5] Mohd Azizi Muhammad nor, Helmi Rashid, Wan Mohd Faizul Wan Mahyuddin, Mohd Azuan Mohd Azlan, Jamaluddin Mahmud "Stress analysis of a low loader chassis" Elsevier Science International Symposium on Robotics and Intelligent Sensors 2012 (IRIS 2012).

[6] Basia PR, Vyas PB, Prof. Patil MY "Dynamic analysis and optimization of heavy motor vehicle chassis frame" International Journal of Engineering & Science Research(IJESR), ISSN: 2277-2685, Vol-3, Issue-5, 3018-3022, May 2013.

[7] Monika S. Agrawal, Md. Razik "Finite Element Analysis of Truck Chassis" International Journal of Engineering Sciences & Research Technology (IJESRT), ISSN: 2277-9655, Dec 2013.

[8] Hemant B. Patil, Sharad D. Kachave, Eknath R. Deore "Stress Analysis of Automotive Chassis with Various Thicknesses" IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), ISSN: 2278-1684 Vol-6, Issue-1, Mar - Apr 2013.

[9] Hemant B. Patil, Sharad D. Kachave "Effects of Web and Flange thickness on bending stiffness of chassis" International Journal Of Advance Research In Science And Engineering (IJARSE), ISSN 2319-8354, Vol-2, Issue-7, July 2013.

[10] Swami K. I., Prof. Tuljapure S. B. "Analysis of ladder chassis of Eicher 20.16 using FEM" IOSR Journal of Applied Geology and Geophysics (IOSR-JAGG), ISSN: 2321–0990, Vol-2, Issue-1, Jan 2014.

[11] Vishal Francis, Rajnish Kumar Rai, Anup Kumar Singh, Pratyush Kumar Singh, Himanshu Yadav "Structural Analysis of Ladder Chassis Frame for Jeep Using Ansys" International Journal Of Modern Engineering Research(IJMER), ISSN: 2249–6645, Vol- 4, Issue-4, April 2014.

[12] Salvi Gauri Sanjay, Kulkarni Abhijeet, Gandhi Pratik Pradeep, Baskar P. "Finite Element Analysis of Fire Truck Chassis for Steel and Carbon Fiber Materials" International Journal of Engineering Research and Applications, ISSN: 2248-9622, Vol-4, Issue-7, July 2014.

[13] V. Vamsi Krishnam Raju, B. Durga Prasad, M. Balarama Krishna, Y. Srinivas "Modeling and Structural Analysis of Ladder Type Heavy Vehicle Frame" International Journal of Modern Engineering Research (IJMER), ISSN: 2249–6645, Vol-4, Issue-5, May 2014.