

A Review: Design and Weight Optimization of YJ3128 Type Dump Truck's Frame

Dharmendrasinh Parmar¹, Divyesh Morabiya²

¹M.Tech. Student CAD/CAM, C.U.Shah University, wadhwan 363030, India

²Professor, C.U.Shah University, wadhwan 363030, India

Abstract— To studies the YJ3128-type dump truck's sub-frames, for the fatigue crack occurred in the Sub frame witch has worked in bad condition for some months, the truck's working conditions and the load of 16tonne features are researched, and ANSYS is used to analyze the stress of the sub-frame. To reduce weight and cost by changing section of frame and materials and also select best section of frame and material. The reason for the cracking of the frame is found, and the improving method. A theoretical basis is provided for the improvement of the frames' designing, and an important reference is provided to improve the truck frame's design methods.

I. INTRODUCTION

The demands for heavy-duty trucks are also multiplying, annual demand of heavy-duty trucks is 95 million or more, and its annual growth rate is 15% or 20%. The mining dump truck runs all the year round in the terrible mine road, which is prone to need higher performances than the general highway vehicle, such as stiffness, strength, and fatigue life. As the main part of the dump truck, the fatigue life for frame is focused on, especially when It is fully loaded. Actually, dynamic forces caused by the road surface roughness are the foremost factor to lead to fatigue failure of the frame during the dump truck services.

However, in general, the stress level of the frame does not exceed the fatigue strength of the material except the local stress concentrations. research on structure fatigue life of dump truck is extremely rare by tests because of its hugeness in size and weight. it is formed by the main-frame, sub-frame, cab, cargo box, the hydraulic lifting mechanism, engine, and gears. the sub-frame connected the main frame and the container, and increased strength and stiffness of the main frame, and lead to the main frame 's distributed load force.

Therefore, the designing of the sub frame has the important influence to the main frame's quality and service life. Heavy trucks is commonly used transport vehicles in the site, mining and other places, their working conditions are bad, and dynamic load is great. The YJ3128-type dump truck is a Transport vehicles designed. For the frame is their important load component, the quality of the frame is directly related to the performance of heavy trucks.

II. LITERATURE REVIEW

A. *Hirak Patel, Khushbu C. Panchal and Chetan S. Jadav*

Automotive chassis is an important part of an automobile. The chassis serves as a frame work for supporting the body and different parts of the automobile. Also, it should be rigid enough to withstand the shock, twist, vibration and other stresses. Along with strength, an important consideration in chassis design is to have adequate bending stiffness for better handling characteristics. So, maximum stress, maximum equilateral stress and deflection are important criteria for the design of the chassis.

The truck chassis design is done analytically and the weight optimization is done by sensitivity analysis. In sensitivity analysis different cross section are used for stress analysis and we find a 17% weight reduction in the truck chassis. The stress and deformation are also compared for the different cross section [1].

B. *Hemant B.Patil, Sharad D.Kachave, Eknath R.Deore*

Stress analysis of a ladder type low loader truck chassis structure consisting of C-beams design for application of 7.5 tonne was performed by using FEM. To reduce the expenses of the chassis of the trucks, the chassis structure design should be changed or the thickness should be decreased. Also determination of the stresses of a truck chassis before manufacturing is important due to the design improvement.

The analyses are processed in the static and structural conditions. From comparison for 4mm thickness the highest stress occurred is 123.83 MPa by FE analysis and the calculated maximum shear stress is 123.83 Mpa. The maximum displacement of numerical simulation result is 0.288 mm. The difference is caused by simplification of model and uncertainties of numerical calculation and improper meshing.

Hence 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 [2].

C. *Chen Yanhong, Zhu Feng*

Mainly studies the YJ3128-type dump trucks sub-frames, for the fatigue crack occurred in the Sub frame witch has worked in bad condition for 3 to 5 months, the trucks working conditions and the load features are researched, and ANSYS is used to analyze the stress of the sub-frame. According the deferent stress, the reason of the fatigue cracks occurring is researched too. At last an improvement and optimization to the structures of the frame is provided.

For the main reason of the crack is the lack of the sub-frame's torsional stiffness, a beam is added to the front of the sub-frame. And the upper surface of the block slope on both sides of the circular beam is turned into inclined plane, then it and the upper surface of the longitudinal are not in the same plane.

The turning point of the left stringer is also reduced greatly, therefore, there is no big turning transform in the improved frames, and the torsional stiffness is enhanced. No excessive stress concentration exists, and the bearing capacity is greatly improved. Thus it meet the design and use requirements [3].

D. Sandip Godse, Prof. D.A.Patel

The static load analysis of the chassis of TATA ace ex using ANSYS workbench and stress optimization using reinforcement technique of optimization. This has been carried out with limited modifications by adding stiffeners. The necessary design changes required to enhance the load carrying capacity of the vehicle has been recommended successfully.

The existing chassis was analysed by the finite element analysis, the stress levels are found to be 37.04 N/mm² at 21091.5 N load applied.

After modifications, the chassis with suitable reinforcement, increase in thickness, addition of stiffeners, the finite element analysis was carried out, and the stress levels of chassis are found as 22.97 N/mm² which demonstrates that the modified chassis is capable to carry the loads beyond the previous payload [4].

E. Sairam Kotari , V.Gopinath

The analysis of chassis frame for improving its payload by adding stiffener and c- channel at maximum stress region of chassis frame. The FEM analysis has been carried out with various alternatives. The results illuminate the new creative ways for optimum frame design which makes it more sustainable for structural concerns. This paper analyzed the backbone frame for both dynamic and static load condition with the stress deflection bending moment on the tatra chassis frame. The finite element analysis over ansys is performed by considering the load cases and boundary conditions for the stress analysis of the chassis.

The existing TATRA chassis was analyzed by the finite element analysis for installation of the Antenna and Electronic components and the stress levels are found to be 737.3 N/mm². After modifications, the TATRA Chassis with suitable reinforcement, increase in thickness, addition of stiffeners, the finite element analysis was carried out, and the stress levels of chassis are found as 173.38 N/mm², which is less than yield stress 410 N/mm². So The generated shear stresses are less than the permissible value so the design is safe.

From the above Results, it can be concluded that the modified TATRA chassis is capable to carry the loads beyond the previous payload upto 14 tonnes [5].

F. Tushar M. Patel, Dr. M. G. Bhatt and Harshad K. Patel

Truck chassis is the structural backbone of any vehicle. The main function of the truck chassis is to support the components and payload placed upon it. The chassis frame has to withstand the stresses developed as well as deformation occurs in it and that should be within a limit. This paper presents the study of the stress developed in chassis and deformation of chassis frame of EICHER 11.10. The stress and deformation has been calculated for the chassis frame and the FE analysis has been done for the validation on the chassis frame model.

The generated shear stresses are less than the permissible value so the design is safe. The analysis gives maximum shear stress and total deformation which are in desired limit and Comparison of analysis and calculated results [6].

G. Mohd Hanif Mat, Amir Radzi Ab. Ghani

Eco-Challenge' race cars are built for fuel efficiency which means they must be lightweight and have low frictional resistance. However they are still subjected to normal car loading such as engine and driver weight, acceleration, braking and cornering forces. The challenge is to develop a lightweight chassis that can safely withstand the required loads. FEA was used to determine the strength and rigidity of the chassis subjected to the required loads.

In shows during deceleration, the engine tends to move forward, hence causing high stress at the mounting points. Displacement distribution is similar as the previous condition where the maximum displacement is located at the seat mounting points. Adding traverse members to the existing seat mounting members will reduce the displacement. The chassis is able to support all the loading conditions and therefore comply with the rules and regulations. In most loading conditions, maximum stresses are well below the yield stress [7].

H. Mohd Azizi Muhammad Nora, Helmi Rashida, Wan Mohd Faizul, Wan Mahyuddinb, Mohd Azuan, Mohd Azlanc and Jamaluddin Mahmuda

Aims to prepare model, simulate and perform the stress analysis of an actual low loader structure consisting of I-beams design application of 35 tonne trailer designed in-house by Sumai Engineering Sdn. Bhd, (SESB). The material of structure is Low Alloy Steel A 710 C (Class 3) with 552 MPa of yield strength and 620 MPa of tensile strength.

Stress analysis using finite element method was successfully carried out to determine high stress value, maximum deflection and its location on a low loader chassis structure. The results of analysis revealed that the location maximum deflection and maximum stress agrees well with theoretical maximum location of simple beam under uniform loading distribution. The generated shear stresses are less than the permissible value so the design is safe. This result use for new design generating with new section for reduce stress and deflection in chassis [8].

III. MATERIAL PROPERTIES AND THE STRUCTURE OF THE SUB-FRAME

I. Material properties

TABLE 1 MATERIAL PROPERTIES

NO.	Material properties	
	Material	16 MnL
1	Young's Modulus	210 GPa
2	Poisson ratio μ	0.3
3	Density	$7.8 \times 10^{-3} \text{ g/mm}^3$
4	Mini. yield strength	345 MPa
5	Mini. tensile strength	510 MPa
6	Max. tensile strength	610 MPa
7	Safety factor	1.2
8	Material allowable stress	287.5 MPa

II. THE STRUCTURE OF THE SUB-FRAME

The YJ3128-type dump truck's frame is formed by the main-frame and the sub one, and they are all edge beam type. The vertical beam of the sub-frame is right up the main frames', and they are connected by the U-bolts. The sub-frame of the truck is developed by the Co., and it's length is 4.7m, forth-width is 0.901m, and the back-width is 0.761m. The 8mm thick V-beam is trough, all its section is the same. There is a trough auxiliary beam flipping in it. There are six beams in the sub-frame, that are a cylindrical beam at the back of it, a square beam in the middle, and four trough beams, Which are showed in figure 1.

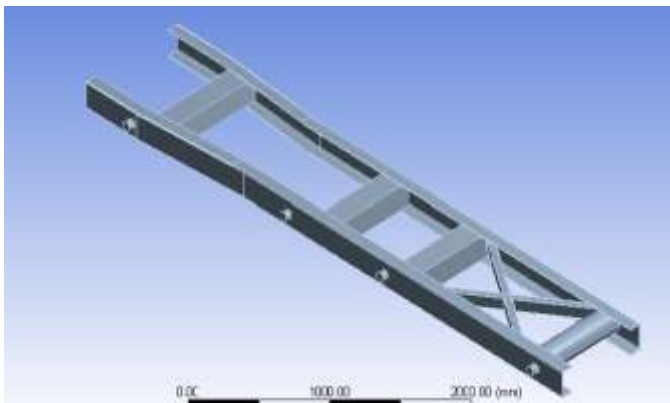


Fig. 1 Truck's chassis/frame

IV. CONCLUSIONS

Heavy trucks is commonly used transport vehicles in the site, mining and other places, their working conditions are bad, and dynamic load is great. For the frame is their important load component, the quality of the frame is directly related to the performance of heavy trucks. The sub-frame is analyzed

by ANSYS, and the reason for the cracking of the frame is found, and the improving method The future scope for research purpose is to make a model of existing bearing in the CAD software and then analyze it on the same. Then to check the results of theoretical and the software. Accordingly have to find out the changes that can be done to the design and make a new model with the changes. It will be again calculated with the theoretical part and then analyzed on the software so that the results can be compared.

REFERENCES

- [1] Hirak Patel, Khushbu C. Panchal, Chetan S. Jadav "Structural Analysis of Truck Chassis Frame and Design Optimization for Weight Reduction" Volume-2, Issue-4, April 2013, Sadvidhyamandal Institute Of Technology, Bharuch, Gujrat.
- [2] Hemant B. Patil, Sharad D. Kachave, Eknath R. Deore, "Stress Analysis of Automotive Chassis with Various Thicknesses" ISSN: 2278-1684 Volume 6, Issue 1 (Mar. - Apr. 2013), Dhule North Maharashtra University, India.
- [3] Chen Yanhong, Zhu Feng, "The Finite Element Analysis and the Optimization Design of the Yj3128-Type Dump Truck's Sub-Frames Based On ANSYS " 2011, Mechanical and Electrical Engineering Institute of Kaifeng University , Kai Feng 475004, China.
- [4] Sandip Godse, Prof. D.A. Patel, "Static Load Analysis Of Tata Ace Ex Chassis And Stress Optimisation Using Reinforcement Technique" Volume4 Issue7- July 2013, Mech. Dept, SPCE, Visnagar, Gujarat.
- [5] Sairam Kotari , V. Gopinath, "Static And Dynamic Analysis On Tatra Chassis" Vol.2, Issue.1 QIS College Of Engineering & Technology Ongole, Andhra Pradesh.
- [6] Tushar M. Patel, Dr. M. G. Bhatt and Harshad K. Patel, "Analysis and validation of Eicher 11.10 chassis frame using ANSYS", Volume 2, Issue 2, March – April 2013, LDRP-ITR, Gandhinagar, Gujarat, India.
- [7] Mohd Hanif Mat, Amir Radzi Ab. Ghani, "Design and Analysis of Eco Car Chassis", IRIS 2012, Universiti Teknologi MARA, Shah Alam, Malaysia.
- [8] Mohd Azizi Muhammad Nora, Helmi Rashida, Wan Mohd Faizul, Wan Mahyuddinb, Mohd Azuan, Mohd Azlanc, Jamaluddin Mahmuda, "Stress Analysis of a Low Loader Chassis", IRIS2012, Universiti Teknologi PETRONAS, Tronoh 31750, Malaysia.