IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684, p-ISSN: 2320-334X
PP 24-27
www.iosrjournals.org

The Failure Analysis of Tractor Trolley Chassis An Approach using Finite Element Method - A Review

Ms. Kshitija A. Bhat[1], Prof. Harish V. Katore[2]
[1]*M. Tech Student of CAD /CAM , Dept of Mechanical Engineering ,D.M.I.E.T.R , Wardha, India*
[2]Dept of Mechanical Engineering, D.M.I.E.T.R , *Wardha* , India

ABSTRACT : *Tractor Trolleys are very popular and cheaper modes of goods transport in rural as well as urban area. But these trolleys are manufactured in small scale to moderate scale industry, due to which design of chassis is at primary level. As design is not properly done according to the stress analysis, so if any type of failure occurs then we can't afford the replacement of the chassis. As the chassis design is not proper and excess material is used to overcome the failure, the self weight of the chassis get increases and capacity of the trolley get decreases. Due to which, the cost of the trolley also get increases. Due to which, the self load of trolley applied on the chassis, leaf spring and axle of the trolley causes failure in dynamic condition. Failure such as breakage of axle hub assembly, breakage of leaf spring, axle bending occurs. And these problems are indirectly related to the self weight of chassis and capacity of the trolley. To overcome these problems, a proper redesigning of chassis according to the stress strain analysis is required by reducing self weight of trolley.*
Keywords: *Tractor Trolley Chassis, Finite Element Analysis, stress Analysis.*

I. NTRODUCTION

1.1 Tractor Trolley:

Trolleys are widely used for transporting agriculture product, building construction material, and industrial equipment. The varieties are available and use of particular trolleys depends upon the application. The main requirements of trolley manufacturing are high performance, easy to maintain, longer working life and robust construction. In this work, tractor trolley which is used for the agriculture work and sometimes used for transporting building construction material is considered. These trolleys are divided into two types as two wheeler and four wheeler. The tractor trolleys are available in various capacities like 3 ton, 5 ton, 8 ton.

1.2 Chassis

A chassis is one of the key components of the vehicle. A chassis consists of an internal framework that supports the container of tractor trolley in its construction and use. It is a dead vehicle which is connected to the tractor to carry the load. The Chassis is used to support the container on which the load is to be carried out.The trolley chassis main frame is supported at two points over the axle. The chassis is connected to wheel axle through isolator leaf spring and to the tractor coupling. The leaf spring attached to main frame at two point where leaf springs are attach to transfer the total load to the axle.

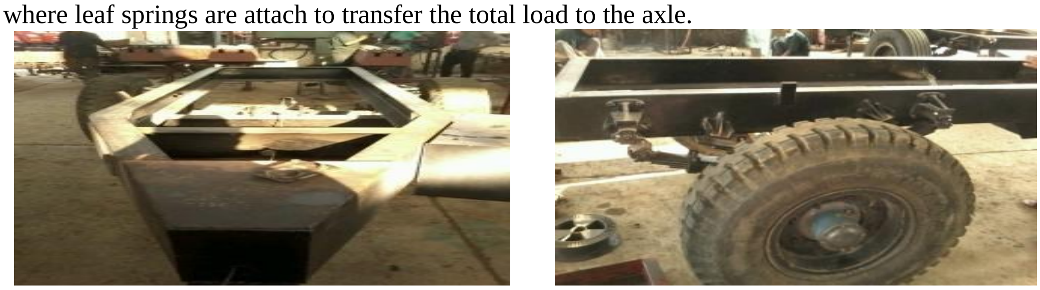

Fig 1.1 Trolley chassis views

II. RELATED WORK

Hemant B.Patil, Sharad D.Kachave [1] presents, 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. The commercial finite element package CATIA version 5 was used for the solution of the problem. 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. In order to achieve a reduction in the magnitude of stress at critical point of the chassis frame, side member thickness, cross member thickness and position of cross member from rear end were varied.

Numerical results showed that if the thickness change is not possible, changing the position of cross member may be a good alternative. Computed results are then
compared to analytical calculation, where it is found that the maximum deflection agrees well with theoretical approximation but varies on the magnitude aspect. 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. Comparing case 5 with case 2 only by increasing thickness of cross member weight is increases by 2.92 Kg stresses are decreased by 43.8 N/mm2, and displacement is increased by 0.022mm. Comparing case 4 with case 2 only by changing the position of cross member weight is not affecting stresses are deceases by 06.7 N/mm2, and displacement is increased by 0.026 mm. Comparing case 3 with case 2 only by increasing 23.58 Kg stresses are increased by 06.5 N/mm2, and displacement is increased by 0.024mm.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.
Roslan Abd Rahman, Mohd Nasir Tamin, and Ojo Kurdi [2] investigated stress analysis of heavy duty truck chassis using finite element method .The result shows that the critical point of stress occurred at opening of chassis which is in contacted with the bolt. Thus, it is important to take note to reduce stress magnitude at this point. The location of maximum deflection agrees well with the maximum location of simple beam loaded by uniform distributed force.

N.K.Ingole, D.V.Bhope [3] does the modifications in existing model of tractor trailer chassis by 1) Variation in Cross sectional areas of cross members 2) Variation in cross sectional areas of cross and longitudinal members, 3) Variation in cross sectional areas of cross and longitudinal members and Changing the position of cross members of main frames of chassis 4)Considering Variable cross sectional areas of cross and longitudinal members. It has been found that, maximum stress present in existing chassis is 75 MPa and weight of chassis is 751.82 kg. Case 4 leads to maximum weight reduction of approx 112 kg as compared to case 1, 2 and 3. So modifications as per case 4 are also recommended, case 3 the weight reduction is 88 kg with maximum stress level in range of 25mpa to 66 mpa.

Cicek Karaoglu et al [4] 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.

Mohd Azizi Muhammad Nor et al [5] performs the stress analysis of an actual low loader structure consisting of I-beams design application of 35 tonne trailer. He uses modeling software CATIA V5R18. 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 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. The maximum stresses and maximum deformation in the chassis of trailer.

J. Gadus [6] used optimization procedure for a mass optimization of a welded framework
of a special tractor trailer designed for transport of seeding machines. This work reports on the basis of the optimization procedure more than 35% savings of the material mass has been achieved.
S.Sane, G.Jadhav, and Anandaraj.h [7] carried out stress analysis on light commercial vehicle chassis using iterative procedure for reduction of stress level at critical locations.

III. FINITE ELEMENT ANALYSIS OF EXISTING CHASSIS

For the FE Analysis, it is necessary to create a solid model of chassis and also to create a FE model. In the present work, static analysis has been carried out for the chassis considering sudden load effects. The chassis of Tractor trailer integrates the main components of system such as rear table, main frame, and extension frame. It is considered that extension frame is integral with main frame of chassis which carries the weight of side panel and pay load acting on top plate. Main frame of chassis is used to mount rear table at the end, which is used for attachment of leaf spring suspension. The front part of main frame is used for attachment of circular plate. Main frame is composed of longitudinal members and cross members as shown in figure 3.1. PRO-E is used for modeling the chassis of tractor trailer. The model has a main frame and extensions frame with length 3750 mm and width 1890 mm.

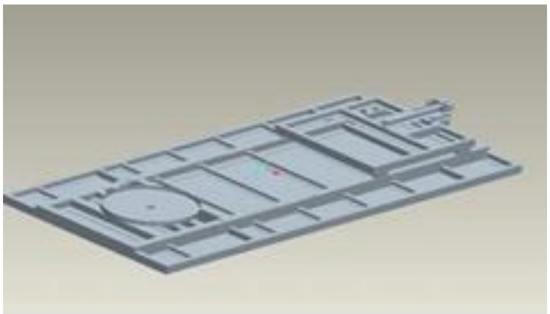

Fig 3.1 CAD model of chassis frame using PROE software

The total weight of trailer including laden and unladed weight is 9000 kg. This load is considered to be distributed over the top plate area of 1890 x 3750 mm2 .Thus, the distributed load considered for analysis is 0.0126 N/mm2. The trailer is also subjected to tractive force which is given by

$F = G \times (\sin \theta + r)$. - (1)

Where, r is rolling resistance of 0.26 for sand surface. Thus tractive force is 79461 N. The properties of Structural steel material of trailer are as shown in table 1. The model is discretized using solid 187 elements with 52443 elements and 106801 nodes as shown figure 3.2.

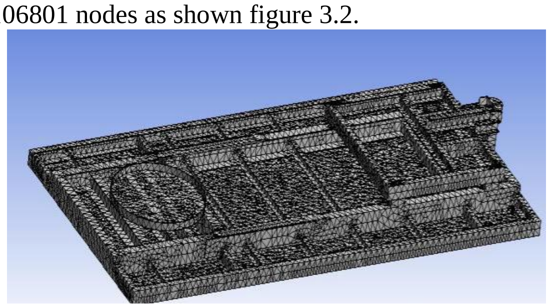

Fig 3.2: Finite element meshing of chassis frame using ANSYS

Supports for tractor trailer chassis are present at rear end of chassis and at rotary table as shown in photograph given in 3.3 respectively. The rear end of the trailer chassis is attached with rear table which is used for attachment of two leaf spring suspensions.

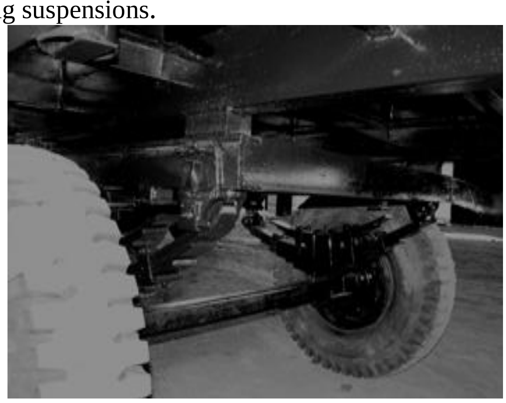

Fig 3.3: Supports for Rear End of Chassis

IV. RESULTS

The FE analysis of existing chassis revealed that, the max stresses are concentrated only over small portion of chassis member. For existing chassis it is seen that maximum stress in longitudinal member is 75 Mpa. If sudden loads effects are considered then stress level may rise to approximately to twice that of static stress of 75 Mpa. Thus the maximum stress will be approximately 150 Mpa. Thus the factor of safety of 1.66 is present for existing chassis. In actual case due to suspensions provided to the chassis, the chassis is subjected to lesser shock loads. The Maximum stress in longitudinal member of extension frame is observed to be 5.78Mpa. Remaining members of chassis are under the minimum stress as shown in figure 4.2. The weight of existing chassis is 751.82 Kg.

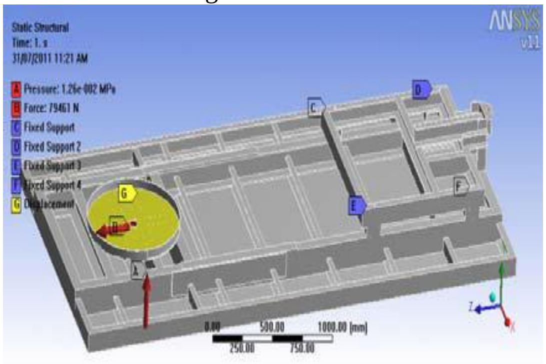

Fig 4.1Boundary conditions, constraints and loading

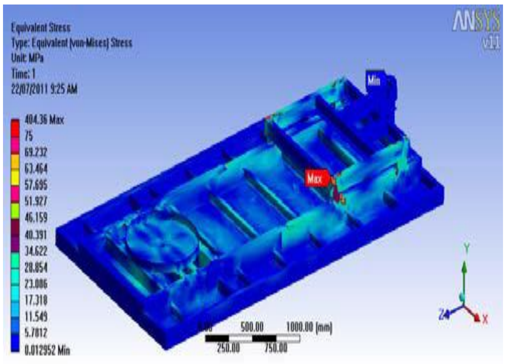

Fig. 4.2 Von Mises Stresses on Chassis

V. CONCLUSION

The finite element analysis shows that the trolley chassis is experiencing various failures. From the overall study of existing chassis, it is seen that whatever failure is occurring is indirectly related to the self weight of chassis .So to overcome these problems, a proper redesigning of chassis according to the stress strain analysis is required by reducing self weight of trolley.

REFERENCES

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), April 2013

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

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.

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 (2002) 1115–1130.

Mohd Azizi Muhammad Nora,, Helmi Rashida, Wan Mohd Faizul Wan Mahyuddin "Stress Analysis of a Low Loader Chassis" Elsevier Ltd. Sci Verse Science Direct Procedia Engineering 41 (2012) 995 – 1001.

J. Gadus, "Optimization of frameworks by means of FEM use" Slovak Agricultural University, Nitra, Slovak Republic.

S.S.Sane, Ghanashyam Jadhav, Anandaraj "Stress analysis of light commercial vehicle chassis by FEM" piaggio vehicles pvt ltd pune.

Patel Vijay kumar V, Prof. R. I. Patel "Structural Analysis of Automotive Chassis Frame and Design Modification for Weight Reduction" International Journal of Engineering Research & Technology (IJERT), ISSN: 2278-0181, Vol. 1 Issue 3, May – 2012.

Tirupathi V. Chandrupatle, Ashok D.Belegundu "Introduction to Finite Elements in Engineering", 6 th edition, 2008, Pearson education, New Delhi.