Stress Reduction of Pickup Truck Chassis Using Finite Element Method

O Kurdi[1, 2, *], M S Yob[1], S R Dasson[1], S Barrathi[1], A A Altayeb[1], and I Yulianti[3,]

[1] Mechanical Engineering Department, Faculty of Engineering, Universiti Selangor, Jalan Timur Tambahan, Bestari Jaya, Selangor Darul Ehsan, Malaysia
[2] Mechanical Engineering Department, Faculty of Engineering, Universitas Diponegoro, Jalan Prof Sudharto, Tembalang, Semarang, Central Java, Indonesia
[3] Physics Department, Faculty of Mathematics and Natural Sciences, Universitas Negeri Semarang, Kampus Sekaran, Central Java, Indonesia

*Corresponding author: ojokurdi@unisel.edu.my

Abstract. This paper performed stress reduction of the chassis of single cab pick truck using Finite Element Method (FEM). The first step was to evaluate and identify the critical area or maximum values of the static stress and its locations. The second stage of this project was dealing with designing a new chassis based on the Tata model, and then the final stage was a validation of numerical results with both experiments and analytical calculation to get the desired chassis that have strong and high-quality design.

1. Introduction

The pickup truck is one of the most important automobiles in the daily life all over the world. It is known for its use on tackling the difficulty of driving on damaged roads as well as in the rough desert land. One of the important parts of a pick up truck is chassis. The chassis of trucks is the backbone of vehicles and integrates the main truck component systems such as the axles, suspension, power train, cab, and trailer. Therefore it is important to have a strong truck chassis that can withstand the weight above it.

Stress is an important parameter of a chassis since high stress may lead to the failure of the chassis. The failure occurs since the magnitude of the highest stress is higher than the magnitude of ultimate stress or yield stress of the materials. There is avarious source of stress that is experienced by chassis which are a static, dynamic and cyclic load.

Numerous researches on thestatic characteristic performance of chassis have been done. Most of the works used FEM to perform stress analysis by employing commercial software such as NASTRAN, ABAQUS and ANSYS [1- 11]. In 2009, V. Veloso et al. [12] studied static characteristic to analyze the default of a longitudinal girder of a vehicle model and then optimized the prototype. The first step in the study was performing durability test on a prototype vehicle experimentally. The test result shows that failures occurred at the fenders fixation region of the suspension system on the vehicle. To overcome the problem, various reinforcement models were proposed, and the performance was analyzed numerically using finite element method (FEM). Modification of existing chassis was done by change the cross section of the chassis [2, 3, 8, 13, 14], reducing the thickness [1, 15] and modifying some part or region [7, 12, 15, 16, 17]. Validation of the simulation results was done by using analytical calculation [18-22]. However, this work proposed new design by change the certain

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd

region of the chassis, reduce the thickness and validate the simulation result by both analytical calculation and also experimental work.

This paper is focused on stress reduction of pickup chassis by modifying the design of the existing model which is Tata Telco line single cab (TATA TL SC). The first step has analyzed the stress due to static load using FEM to evaluate and identify the maximum stress values and its location. After the identification was performed on the existing model, then modification was done to reduce the magnitude of stress in the region. The last step was a validation of FEM simulation result of the modified model by using the experimental work and analytical calculation.

2. Method

The existing chassis type is a ladder frame and was depicted in Figure 1. It consists of two long sections or beams. One section is on the right side, and the other is symmetrical to that on the left side. These beams are called the longitudinal members where all load will be distributed to. There are various types of cross-sectional members that commonly used which are *I*, *C*, and rectangular beams. In TATA TL SC, the rectangular beams are used. There are other members called cross members which connect the longitudinal members together by welding or bolt. These cross members are used to help the longitudinal members to get good strength and stability. Rod members or normal plats are used for this type of members. The longitudinal members are upswept on the front and the rear sides of the frame to provide space for the axils movement caused by the springs.

In this study, the stress analysis is accomplished by the commercial finite element packaged ABAQUS. The simulation was started by assigning the material properties on the model, putting the load and choosing the boundary condition. The material of chassis is Steel with 552 MPa of yield strength and 620 MPa of tensile strength.

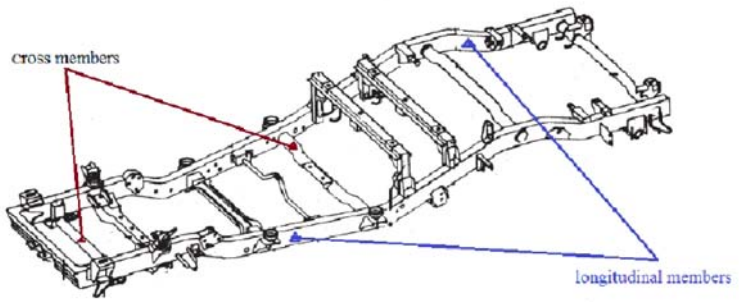


Figure1. TATA TL SC chassis frame (single cab, 4 x 2 and 4 x 4), (Ladder frame). (Tata Motors Limited)

3. Results and Discussion

Simulation result for the existing chassis is shown in Figure 2. The maximum Von Misses stress at the critical region is 227.2 MPa. For failures analysis, the magnitude of stress was compared to the ultimate tensile strength of the material. If the stress exceeds the ultimate stress of the material, the structure is considered as failed. The stress value obtained above are below yield stress. The safety factor for this condition is 2.42. Based on the data from the Department of Structure and Transport of Australian (Department of Structure and Transport), it was recommended by the manufacturer of trailer structure that their producthas the safety factor of at least 3 for highway usage and 5 for off road usage. The truck chassis also experience cyclic loading from road roughness. Structure with high stress which experiences cyclic loading will have alow life time. Therefore, it is important to reduce the maximum stress to improve the chassis life time. The stress can be reduced by optimizing the geometry in which the maximum stress occurred or by increasing the thickness of chassis.

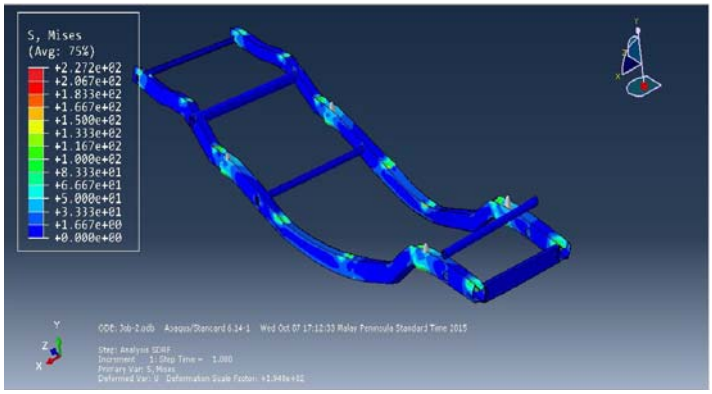


Figure 2. Von Misses stress of the existing chassis model

The modification was done by adding some cross members into the model to reduce the stress on the chassis (design 1) as shown in Figure 3. The cross section shape of the cross members also has been changed from the circular shape to the square shape. It will increase the stiffness and also inertia moment of structure that further results in stress reduction on the chassis. The simulation result of the modified model in term of Von Misses stress is shown in Figure 3. The Von Misses stress of the modified model was decreased by 23.42 % . The safety factor for this condition is 3.16 and the requirement for safety limit is satisfied.

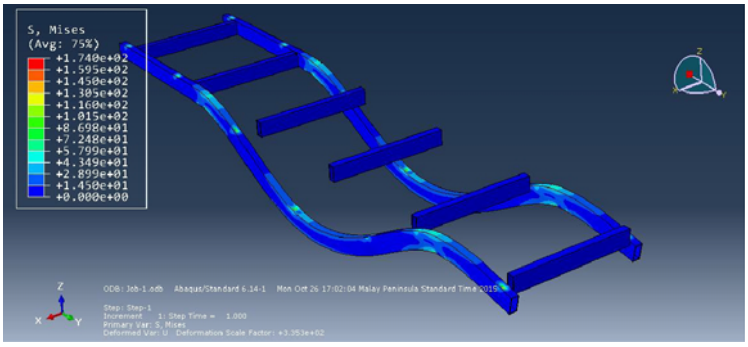


Figure 3. Von Misses Stress Of Design 1

Another modification was done to reduce the stress by proposed new design (design 2) as shown in Figure 4. The new proposed model contains two longitudinal members and six cross members. Both cross member and the longitudinal member has the square hollow shape. The model has the same type with the existing, but it is simpler since some complex shape in the existing design has been removed. The simulation result of design two at the front part of the chassis is shown in Figure 5. The magnitude of Von Misses stress at the front is 46.9 MPa that is much lower than that of the existing and the modified model. The design 2 shows the best performance compared to that of the existing and design 2.



Figure 4. Design 2 of the proposed model

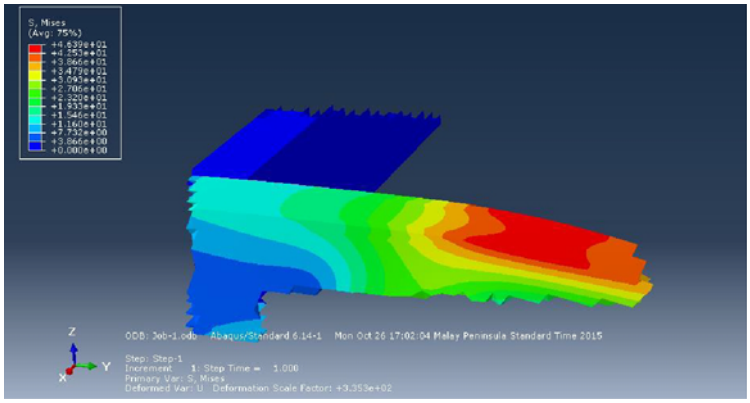


Figure 5. Von Misses stress for design two at the front part.

For validation purpose, the simulation results of the existing model were compared to the analytical calculation, while the simulation results of design two were compared to experimental results.

For analytical calculation, the structure can be assumed as simple beam subjected to several single loads. The bending moment distribution can be calculated after all of the reaction forces were obtained. Figure 6 shows the bending moment distribution of the existing model.

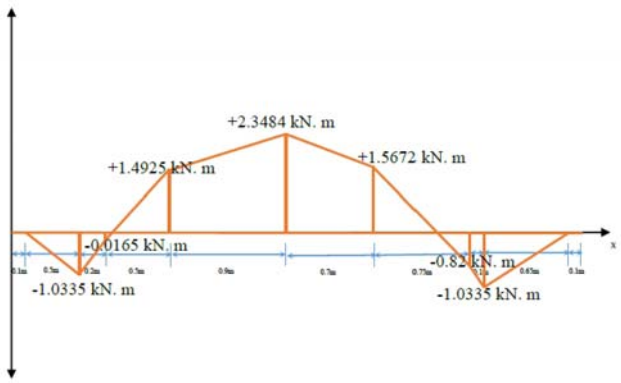


Figure 6. Bending moment diagram for the existing model

The value of Von Misses stress can be calculated using the formula for principle stress and Von Misses stress. The Von Misses stress obtained from the calculation is 240.5 MPa. The percentage different between simulation and analytical for existing model is 5.53%. It shows that the simulation result for design 1 is valid.

Comparison of the experimental results and the simulation results of design two is shown in Table 1. The comparison was made by comparing the Von Misses stress value at three locations which are point 1 at there are part, point 2 of the middle part and point 3 at the front part.

TABLE 1. Von Misses stress result comparison of design 2

	Von Misses stress (MPa) and point location		
	1	**2**	**3**
FEM result	5.9	17.6	29.2
Experiment result	12.43	21.5	46.5
Percentage different (%)	52.53	18.14	37.20
Average			**35.96**

Based on the percentage different on Table 1, it can be seen that the percentage different is quite high. The high different is due to assumption and simplification that was made in thesimulation. Meanwhile, the accuracy of the experimental results also depends on the setting of equipment, accuracy of the sensor and stability of the surrounding condition such as temperature and humidity. Some errors in measurement during the experiment most probably happened. However, the average percentage different of 35.96% can be accepted.

4. Conclusions

The Finite element method has been used to perform stress analysis of the chassis of single cab pick truck "Tata Telco line single cab" (TATA TL SC) successfully. The critical region of the chassis can be identified which was indicated by the highest Von Misses stress value. The modified models with better performance have been proposed. The modified model has asafety factor of 3.1 that higher than the required safety factor (3). The simulation results of the modified models are valid since they meet both analytical and experimental results.

Acknowledgement

We would like to thank Universiti Selangor (UNISEL) for funding the research. Our gratitude also goes to members of structure and materials laboratory, Civil Engineering Department UNISEL for providing the facilities for experimental work and the members of Mechanical Engineering Department UNISEL for their helpful discussion throughout the completion of this work.

References

[1] Agrawal M S & Razik M. 2013. *Int. J.Eng. Sci. Res. Technol*. 2(12), 3432-3438.
[2] Bhat KA, Untawale.SP, & Katore.HV. 2014. *Int. J. Pure. Appl. Res. Eng. Technol.***2(12)**71.
[3] ChandraM R, Sreenivasulu S& HussainS A. 2012.*Int. J. Model. Eng. Res*, **2(4)**2594.
[4] Rajappan R&Vivekanandhan M. 2013.*Int. J. Eng Sci*.**2(2)** 63.
[5] Dubey A & DwivediV.2003.The Vehicle Chassis Analysis by Load Cases & Boundary Conditions for Stress Analysis.
[6] R. Choudharya, T J. Bowsera, P. Wecklera, N. O. Manessb, and W. McGlynnb, 2009 Postharvest. Biol. Tec.**52**103.
[7] Si G M, Sun H, Zhang Q L, &Duan L. 2011b. *Appl. Mech. Materil*.**55-57**. 110.

[8] Naveen Ala, K.Tejdeep Reddy , BVSS Bharadwaja. 2016. I*nternational Journal of Innovative Reesearch in Science, Eng. and Tech.***5**326-337
[9] Rahul.V, Nirmal.R.U, Noraje Aniket Arunkumar, P.Baskar. 2016. *Int. Journal of ChemTech Research***9**384-391
[10] Gaurav Goel, Rajat Garg, Tarun Ranjan, Pratyanshu Soni and Baskar P. 2016. ARPN Journal of Eng.and Applied Sci. **11** 13599-603
[11] Bhuwanesh Sharma, Abdul Samad, Dr.Y.B.Mathur 2016 Int. Conf. on Emerging trends in Engineering & Management for Sustainable Development (Bikaner, Rajasthan, India)
[12] Veloso V, Magalhães H S, BicalhoG I & Palma E S.2009.Eng Failure Analysis.**16(5)** 1696.
[13] NayakD,Sharma K P& Parkhe A.2014.*Int. J. Adv. Technol.Eng.Sci*, **2(5)**292.
[14] Patil, Kamlesh Y & Deore, Eknath R.2015.*J.Mech.Civil.Eng*. **12(4 Ver. III)** 111.
[15] Patil H B, Kachave S D, Sharad, & Deorem E R.2013.*J. Mech. Civil. Eng.*6(1) 44.
[16] Bhoi A, Koushik L P, YadavN & Patnaik M.2013. Int. J. Eng. Res. Appl.**3(2)**1149.
[17] Patel V V, & Patel R I.2012. *Int. J. Eng. Res. Technol*. **1(3)** 1.
[18] Patel T M., Bhatt MG & Patel HK.2013. *Int. J. Emerg. Trend. Technol. Comput. Sci*.**2(2)** 85.
[19] PatilN R, Kulkarni R R., Mane B R & Malve S H. 2014.**3(5)**661.
[20] Bajwa M S, Yatin R&Joshi A. 2013. *Int. J. Mech. Prod Eng.* 1(2).55.
[21] Sharma, P.K, Parekh, Nilesh J., & Nayak, Darshit.2014. *Int. J. Innov.Res.Sci.* 5(8). 182.
[22] Gaikwad A V & Ghawade P S. 2014.*Int. J. Technol Sci*.**2(6)** 12.