Static Stress Analysis of *Merah Putih* EV Bus Chassis by Finite Element Modelling

Muhammad Patrioda, Abdul Munir Hidayat Syah Lubis(✉), and A. Yulianto

Department of Mechanical Engineering, Universitas Muhammadiyah Surakarta, Jalan Ahmad Yani Tromol Pos 1, Pabelan, Kartasuram, Surakarta, Indonesia
{amh612,ay160}@ums.ac.id

Abstract. Emission-free electric vehicles have been well accepted as an environmentally friendly vehicle nowadays. Electric buses are considered as potential future mass transit vehicles that have special advantages to the environment compared to conventional internal combustion engine buses. Indonesia has issued two regulations on the utilization of Electric vehicles and is currently developing a national electric bus project. Chassis design is one of the most important tasks for the vehicle designer to ensure the safety and comfort of the vehicle. Structural analysis by finite element method has been proven to numerically predict stress distribution that affects structure of vehicle chassis. This work is meant to evaluate stress distribution on chassis of Merah Putih bus due to static loading. The static analysis was performed by finite element analysis with the help of Ansys static loading solver. CAD model of actual chassis developed to represent actual chassis of Merah Putih bus. The numerical simulation results showed that maximum total deformation of 5.3957 mm was located on the rear side of the bus where the battery compartment segment is located. Maximum stress of 140 MPa observed at driveline and main frame connection which has the lowest safety factor of 1.5481. Based on these numerical simulation results, the chassis design is considered safe to be used.

Keywords: Finite Element Modelling · Stress Analysis · *Merah Putih* Bus · Chassis

1 Introduction

The role of mass transportations in urban areas are significantly important to support the citizen activities that are in line with the increasing population. The use of buses as mass transportation in urban areas is still a favorite compared to other type of vehicles. However, with growing concern on the effect of vehicle emission on the environment, the utilization of conventional internal combustion engines has become concern nowadays. This concern related to the fact that incomplete combustion of diesel engine, which powered most buses, produces NO_2, CO_2, and H_2O which may cause harm to the environment.

T. P. Syawitri et al. (Eds.): MEST 2022, AER 222, pp. 268–275, 2023.
https://doi.org/10.2991/978-94-6463-134-0_25

The advancement of automotive technology demands technologies that are able to provide more efficient, effective, and environmentally friendly vehicles. Both academic and automotive industries have paid close attention to the development of energy efficient and environmentally friendly vehicle technology today. Indonesia's Ministry of Industry has issue two regulations on utilization of electric vehicles through Presidential Regulation Number 55 of 2019 on Acceleration Program for Battery-powered Electric Vehicle for Road Transportation [1]. The utilization of electric buses has become attractive to provide more efficient and less pollution mass transportation.

Various FEA software packages have been employed to study the stress analysis of vehicle chassis. Bus chassis design is one of the most important tasks for the vehicle designer to ensure the safety and comfort of the vehicle. The chassis typically holds the whole vehicle's components systems together such as driveline, suspension, power train, brake system, *etc.* as well as the main body which contains the passengers. In line with the requirement on increasing energy efficiency, lightweight chassis design and material selection are becoming more important.

Stress analysis, either static or dynamic analysis, is important task for vehicle designers. Finite element analysis (FEA) has been considered as common tool for vehicle chassis analysis [2–5]. In electric buses, a lightweight and strong chassis shall increase power output efficiency of the vehicle. Moreover, good selection of strong and high-resistance materials for the chassis can maintain rigidity and flexibility of the vehicle which can be easily modelled and optimized by finite element method. This work is meant to evaluate stress, deformation, and safety factor distribution on the chassis of *Merah Putih* bus due to static loading by finite element modelling.

2 Method

2.1 Solid Modeling and Meshing

In this work, the *Merah Putih* Bus chassis structure was generated by commercial CAD tool and furtherly imported into Ansys Spaceclaim package. The chassis is of ladder frame type and modeled by using actual dimension of the fabricated chassis as shown in Fig 1. The total length of the chassis is 7373 mm and 2067 mm in width. Main parts of the chassis were modeled as 100 PFC C-Beam. Two types of steel, *i.e.* SPA-H ($E = 190$ GPa, $\nu = 0.3$, $S_y = 355$ MPa) and SUS 201 JIS G4303 ($E = 195$ GPa, $\nu = 0.3$, $S_y = 275$ MPa) were assigned to the chassis model as shown in the Fig 1. The SPA-H material was assigned for driveline frame segments of which connecting the driveline and main frame and SUS 201 JIS G4303 material was assigned for the main frame of chassis. Furthermore, the model was meshed with SHELL 181 elements and TARGE170 and CONTA174 elements also used to define joints between segments of the chassis. To analyze the accuracy level of the numerical analysis, mesh convergence analysis was performed at three different mesh size.

2.2 Boundary Condition (BC) and Loading

Ansys™ static analysis package was employed to model the loading condition of *Merah Putih* Bus chassis. Fixed constraint was assigned as boundary condition and located at 4

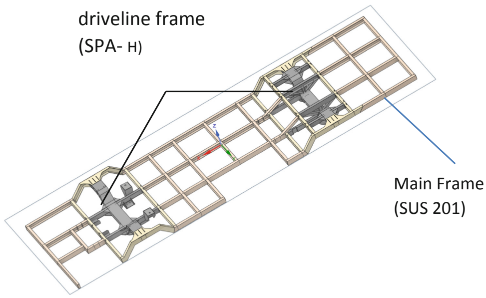


Fig. 1. Solid model of *Merah Putih* bus chassis and material designation

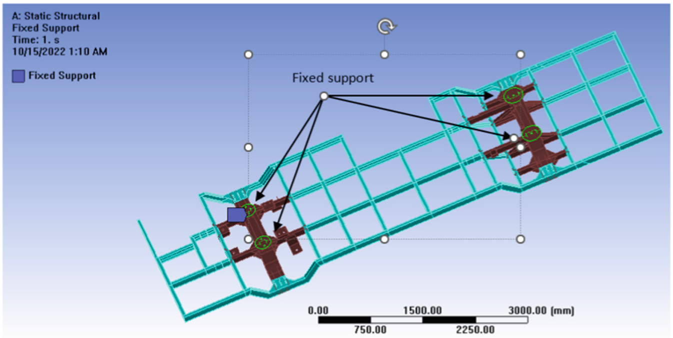


Fig. 2. Boundary conditions

(four) locations at the driveline frame as shown in Fig 2. This BC represents the chassis and driveline assembly supports. The chassis was loaded with different force magnitude at 2 (two) different location, *i.e.,* front side (950 mm length) and rear side (6523 mm length) as shown in Fig 3. Assumption made on the loading is based on the real condition where the front chassis side shall sustain the weight of a driver and front cowl of bus with a total weight of 257.2 kg (F1), while the rear chassis side shall sustain 19 passengers, body of bus and other components attached to the chassis such as motor controllers, battery packs etc. with a total weight of 3307.8 Kg (F2).

2.3 Post Processing

The finite element numerical modelling typically offers various solution results to analyzed. In this study the equivalent stress, deformation, and safety factor were selected results for analysis.

3 Results and Discussion

3.1 Mesh Convergence Analysis

Mesh convergence analysis is an important and commonly used for validating numerical analysis results. Conducting a convergence study shall provide insight into the better

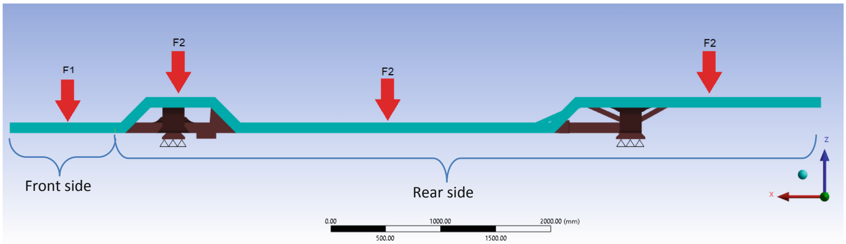


Fig. 3. Loading conditions of the model

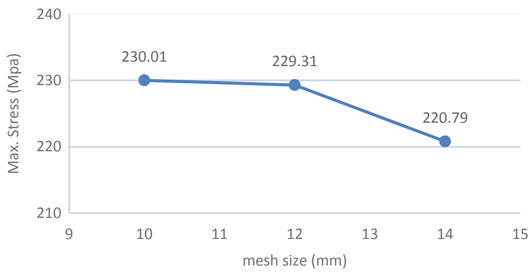


Fig. 4. Mesh convergence analysis

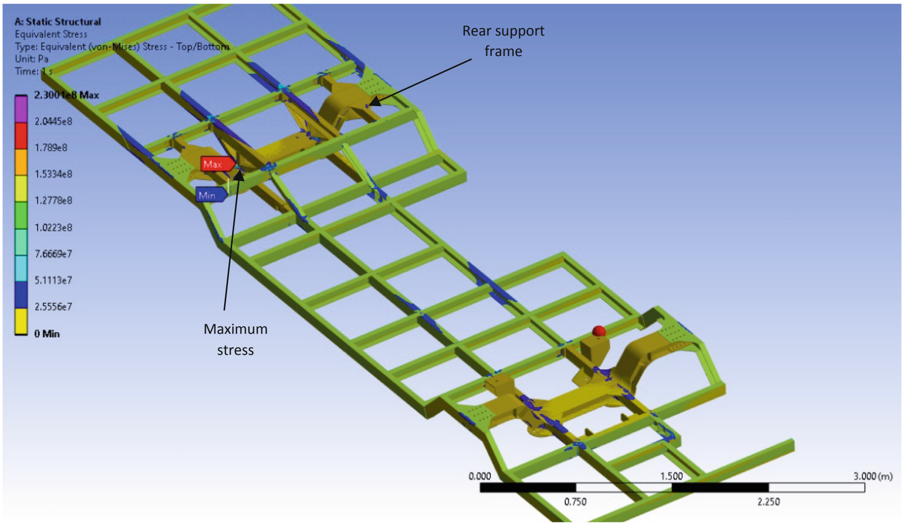
mesh size for more accurate results as well. System response such as stress and deformation typically will converged with decreasing element size with the cost of solution time. In this work, mesh of different size, *i.e.,* 10, 12, and 14 mm respectively, were employed to check over the convergent output result. Figure 4 shows the convergent output results for three mesh sizes over maximum stress on the chassis. The result suggests that a finer mesh provide higher and possibly more accurate maximum stress. No significant result observed from the 10 mm and 12 mm mesh size. Therefore, the simulation is satisfying the convergence criteria.

3.2 Stress Analysis

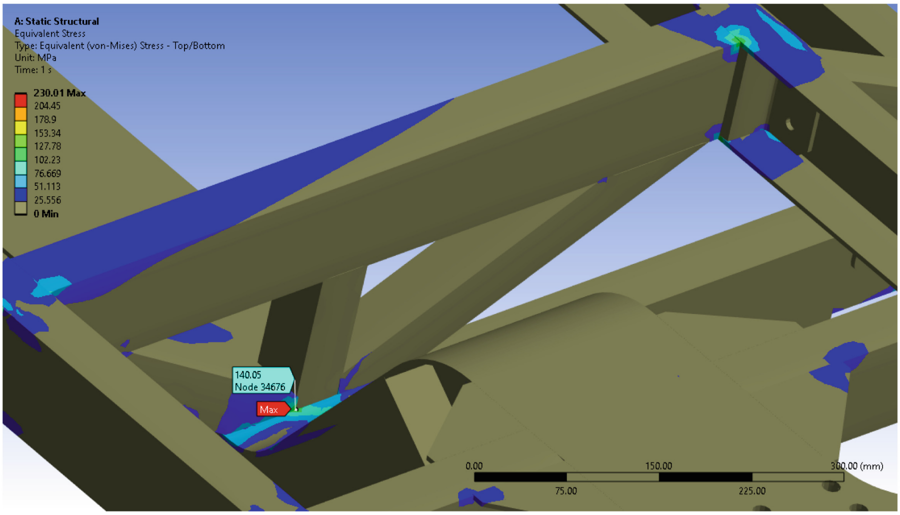
Stress analysis is an important analysis to understand how the load will generate stress to the structure. Figure 5 (a) shows the contour of Equivalent Stress acting on the chassis structure and Fig 5(b) shows the detail on the maximum stress location. High stress observed at several joint connections. The maximum stress of 140 MPa occurs at rear driveline frame which made of SPA-H material. However, this stress magnitude is still allowed ($\sigma < E$).

3.3 Deformation

To ensure vehicle safety, the bus chassis must have enough stiffness to hold whole body components of the vehicle. Deformations can cause the chassis to experience changes in shape which can lead to failure. This property is related to material properties such as



(a)



(b)

Fig. 5. Contour of Equivalent Stress (a) overall view which shows maximum and minimum stress location (b) Detail location of the maximum stress.

material strength and modulus elasticity to sustain both elastic dan plastic deformation that occurs. Deformation that exceeds the permissible stress limit can causing the material to fracture. The fracture of such a heavy vehicle's frame member can be caused by the joints where welding process possibly does not completely melt the joint material [6]. Figure 6 shows deformation contour of the chassis due to the loading. Maximum displacement (5.4 mm) observed at the end of rear side of the bus where the batteries are arranged. Distortion of vehicles members must be kept as low as possible. A deflection

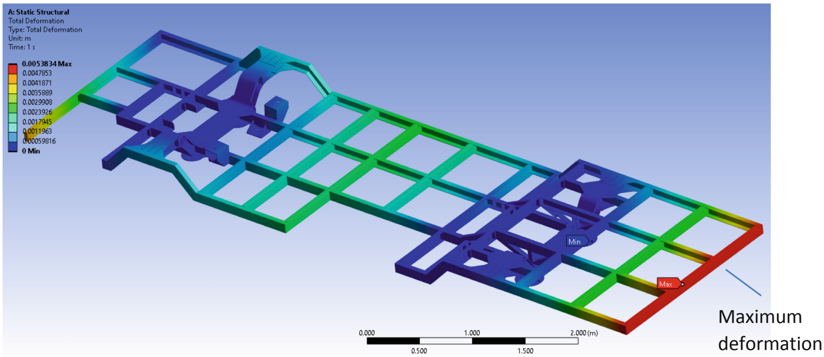


Fig. 6. Contour of Total deformation

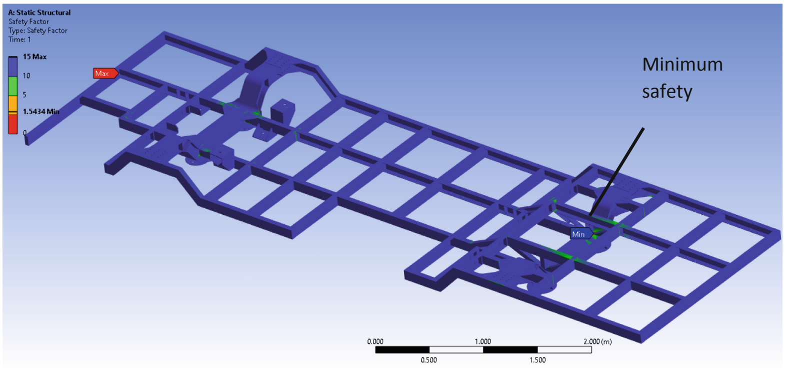


Fig. 7. Contour of Safety Factor

index established empirically stated that the ratio of displacement to chassis span should not exceed 1/240 [7]. In this case, the ratio of displacement to chassis span is much lower than this criterion. Therefore, the deformation is still safe, but it is suggested that reinforcement should be installed on this location to increase the structural stiffness.

3.4 Safety Factor

Figure 7 show the contour of the safety point for safety factor (SF). The lowest safety point (1.5481) is observed at the rear chassis - wheel axle segment made of SPA-H material. This location is also the location of the highest stress as shown in Figure 5 based on the Von Mises criterion to calculate the safety factor. Maximum safety point located at the end of the bus front side made of SUS 201 material which also related to low stress at this location. This result is as expected since this location only sustains lower load compared to the rear side. Moreover, low safety factors also observed at most joints which are typical for welded joints. Therefore, attentions need to be given for these joints.

4 Conclusion

Stress analysis performed on the *Merah Putih* Bus chassis successfully predicts stress distribution, deformation, and safety factor with good mesh convergence. High stress typically observed at joint connections and maximum stress of 140 MPa occurs at rear support where the driveline and main frame are connected. However, the stress is considered as safe since the maximum stress is still under permissible stress value. The chassis frame experiences deformations in which maximum total deformation of 5.4 mm observed at the rear side of the chassis. This maximum deformation is located at the battery compartment segment. However, the value of the total deformation results is considered safe because it does not exceed the permissible limit according to the deflection index. The minimum safety factor with SPA-H material is 1.5481 and SUS 201 is 1.931. From these results it can be concluded that the designed chassis with combination of SPA-H and SUS 201 materials is safe under the specified static load.

Acknowledgement. The authors would like to thank PT Industri Kereta API (INKA), Indonesia, especially Mr. Hernawan Prasanto, for their full support in this study.

Bibliography

1. A. D. Kantor Seskab Bidang Dokumen Negara, "Indonesia Ready to Welcome EV Era, Minister Says," 2022. https://setkab.go.id/en/indonesia-ready-to-welcome-ev-era-minister-says/.
2. N. M. Ghazaly, "Applications of Finite Element Stress Analysis of Heavy Truck Chassis: Survey and Recent Development," J. Mech. Des. Vib., vol. 2, no. 3, pp. 69–73, 2014.
3. D. Croccolo, M. De Agostinis, and N. Vincenzi, "Structural Analysis of an Articulated Urban Bus Chassis via FEM: a Methodology Applied to a Case Study," J. Mech. Eng., vol. 57, no. 11, pp. 799–809, 2011.
4. H. K. Asker, T. S. Dawood, and A. F. Said, "Stress Analysis of Standard Truck Chassis During Ramping on Block Using Finite Element Method," ARPN J. Eng. Appl. Sci., vol. 7, no. 6, pp. 641–648, 2012.
5. T. H. Fui and R. A. Rahman, "Statics and Dynamics Structural Analysis of A 4.5 Ton Truck Chassis," J. Mek., vol. 24, no. December, pp. 56–67, 2007.
6. J. Caban, A. Nieoczym, and L. Gardynski, "Strength analysis of a container semi-truck frame," Eng. Fail. Anal., vol. 127, p. 105487, 2021.
7. F. A. Leckie and D. J. Bello, Strength and Stiffness of Engineering Systems. Springer Science & Business Media, 2009.