STATIC LOADING ANALYSIS ON THREE-WHEELED ELECTRIC VEHICLE FRAME USING SOLIDWORKS

Faruq Avero Azhar[1*], Dwi Djoko Suranto[1], Alex Taufiqurrohman Zain[1], Cahyaning Nur Karimah[1], Dicky Adi Tyagita[1], Aditya Wahyu Pratama[1], Dety Oktavia Sulistiono[1], Mochammad Nuruddin[2], Mochammad Irwan Nari[3]

[1] Program Studi Mesin Otomotif, Politeknik Negeri Jember
[2] Program Studi Teknik Energi Terbarukan, Politeknik Negeri Jember
[3] Program Studi Teknologi Rekayasa Mekatronika, Politeknik Negeri Jember

*Email: faruq.avero@polije.ac.id

ABSTRACT

Electric vehicles are one of the potential solutions to address environmental issues and dependence on fossil fuels in the transportation sector. Therefore, a comprehensive analysis of the electric vehicle frame is very important. Finite Element Analysis had become an industrial standard for designing and analyzing vehicle structures. This study was to analyze the behavior of a three-wheeled electric vehicle frame when the load acting on the chassis using Finite Element Method. Simulation was done to determine the design performance of the chassis using SolidWorks software. The value of stress, maximum strain, and displacement with static loading of the frame structure on three-wheeled electric vehicles presented in this article. The three-wheeled electric vehicle was expected to have a large carrying load capacity, so this simulation is very necessary to predict any damage. The simulation results show that the maximum stress value on the chassis frame when 1500 N given by load is 0.0244 MPa. The minimum and maximum strain values generated were 5.26×10^{-13} and 1.27×10^{-7}, respectively. While the displacement value obtained 6.208×10^{-4} mm. The results of failure analysis with von mises criteria, it can be concluded that the frame design is good enough to withstand the given load.

Keywords: electric vehicle, three-wheeled, failure analysis

INTRODUCTION

Paris Agreement on climate change has triggered an increasing trend of green energy in most countries around the world. Each country is responsible for playing a role in reducing the average global ambient temperature by 2°C [1]. Exhaust emissions caused by the combustion of internal combustion engines contribute to increasing global environmental temperatures. Vehicles with internal combustion engine technology are still widely used in all countries, it is potentially to contribute to the impact of environmental damage and global warming [2]. Meanwhile, the need for vehicles as mass and personal transportation continues to increase. In 2018, commercial vehicles spread around the world reached 380 million and passenger cars approached 1.2 billion [3]. That indicates that people's dependence on transportation is very high, and it is impossible to eliminate its existence [4].

Technological developments and innovation activities in the transportation sector had been continued based on people's dependence on transportation as an effort to support the vision of increasing green energy by reducing the use of fossil energy resources and reducing exhaust emissions. The presence of electric vehicles is the answer challenge to reduce the level of exhaust emissions and global warming [5]. The growth of electric vehicles continues to increase every year because it provides several advantages, including high efficiency, minimal maintenance, environmentally friendly, especially if integrated with renewable energy resources [6]. Ultimately, electric vehicles play an important role in reducing greenhouse gases as an alternative to internal combustion engines [1]. Therefore, it is necessary to expand the use of electric vehicles in the industrial and agricultural transportation sectors to achieve a reduction in global environmental temperatures.

Electric vehicles are one of the potential solutions to overcome environmental problems and dependence on fossil fuels in the transportation sector. Electric vehicles are not only used in personal transportation, but also can be utilized in the agricultural transportation sector. The agricultural sector is both a contributor and an affected sector due to global warming. Production activities in the agricultural sector contributed 11.2% of total global greenhouse gas emissions in 2014, which is 1% of total global greenhouse gas emissions came from the agricultural machinery was used fossil fuels [7]-[9]. Three-wheeled agricultural transport is an attractive alternative in modern agriculture. However, to ensure the performance, safety, and reliability of electric vehicles in agricultural environments, carefully testing is required.

In the scope of three-wheeled agricultural transportation using electric vehicles, the vehicle frame plays an important role in determining performance and durability. Chassis is the place where important components are attached to the vehicle, while supporting all loads (the weight of each component, the forces that arise when accelerating, decelerating, and cornering) making the frame has an important role and has the impact of increasing the weight of the vehicle [10]. Therefore, a comprehensive analysis of the electric vehicle frame is very important. In addition, to minimize the development cost, time, and risk in physical testing, computer simulation can be used as an effective tool.

Finite Element analysis had become an industrial standard for designing and analyzing vehicle structures. Structural analysis is needed to determine if the pressure caused by the load is well distributed. So, it is expected that there will be no high pressure at one point on the chassis frame which will result in plastic deformation when it receives a load, whether loading the driver, engine, body, cargo, or other parts. [11]. This article would analyze the behavior of the three-wheeled electric vehicle chassis when subjected to loads. The effect of the loading applied to the chassis frame was analyzed using the Finite Element Method approach to determine the design performance of the chassis frame that will later be realized.

LITERATURE REVIEW

An analysis using the Finite Element method produces comprehensive and detailed data. The physical response of the system to loads at several points could be read. This is very useful in engineering, especially in vehicle and structural analysis. Guron, et al. (2013) have optimized the bracket component of the cross member on the chassis of a truck. Stress analysis at critical points that cause failure was successfully carried out using the Finite Element method with PATRAN software. Based on the results of the analysis, remodeling is being carried out to reduce the stress value that leads to safety design [12].

Automotive designer was able to understand the various stresses and responses of a chassis design. Stress analysis is important to study in relation to fatigue stress and the lifetime of a system. Ghazaly (2014) has conducted a literature review on finite element methods from 4 different software developer platforms, namely ABAQUS, ANSYS, NASTRAN and HYPERVIEW and used to analyze the frame of a truck. It was observed that the most common basic FEA (Finite Element Analysis) package is suitable for this analysis. Moreover, it was found that most of the researchers used ABAQUS and ANSYS software to predict the stress analysis of the chassis [13].

Hadimani, et al (2018) conducted tests on the chassis frame using several different materials, namely, composite steel, A360 aluminum, magnesium, and carbon fiber. 3D modeling of the chassis frame was drawn using NX-CAD software and test analysis using ANSYS. Structural and modal analysis was successfully conducted with the result that carbon fiber is the best material when applied to the chassis frame compared to other materials [14].

RESEARCH METHODOLOGI

Three-wheeled vehicles are widely used as transportation vehicles in various business sectors. It has the advantage of not too large dimensions, small turning angles, and rear-wheel drive [15]. The configuration of the three-wheeled vehicle has one wheel in front supported by telescopic suspension and two wheels at the rear using differential gears. The chassis frame of this vehicle was made using a hollow shape with cast carbon steel material which is widely used in the automotive, construction and industrial equipment sectors. The hollow shape was chosen because it could withstand bending and torsional forces very well. Welding was done on hollow shape that has been cut according to the desired dimensions to be assembled into a solid chassis frame structure. The overall frame design can be seen in Figure 1. The 3D design was created using SolidWorks Software.

Figure 1. Three-Wheel EV chassis

The chassis of this three-wheeled vehicle is the place where several components are attached, namely, the truck bed, battery, controller and so on. Figure 2 shows the load distribution received by the chassis frame. In this test, the frame structure will be subjected to a load of 1500 N distributed on the frame as shown by the purple arrow. The boundary condition point is located at the leaf spring support and is set equal to zero in the longitudinal and transversal directions. This condition setting is called fixed constrain.

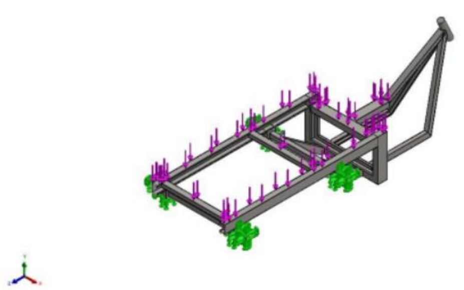

Figure 2. Chassis Load Distribution

The frame material uses cast carbon steel because it is easily to found and easy to weld so that the frame shape would be as well as expected.

Detailed material specifications of the chassis frame are in accordance with Table 1. The mesh in this simulation test had different sizes and automatically generated by Solidworks software, where the meshing results fit the complexity of the frame shape and computer capabilities. The total number of nodals and elements generated were 24,118 and 11,502, respectively.

Table 1. Specification of Three-Wheel EV Chassis Frame

Parameter	Specification
Name	Cast Carbon Steel
Yield Strength	248,2 MPa
Tensile Strength	482,5 MPa
Elastic Modulus	200000 MPa
Poisson's Ratio	0,32
Mass Density	7.800 kg/m^3
Shear Modulus	76000 MPa
Thermal Expansion	1,2x10^{-5} /Kelvin

RESULT AND DISCUSSION

Obtaining an optimum and convincing design of the chassis frame before production, a static structural analysis was conducted. Analysis of the load applied to the chassis frame was obtained through simulation testing using Solidworks software. Figure 3 shows the stress distribution caused by load of 1500 N. Based on the simulation results, the maximum stress on the chassis frame generated by the load is 0.0244 MPa, which is still far below the yield strength of the material which is 248.2 Mpa, so the frame design meets the safe criteria [16]. The maximum stress is located in the center of the frame where the front side leaf spring rests. Analysis of the von mises stress value in the simulation results shows the ability of the structure to withstand the load received. Based on the results of the von mises stress analysis, it can be concluded that the design frame structure that has been made does not fail or fracture when the load is given.

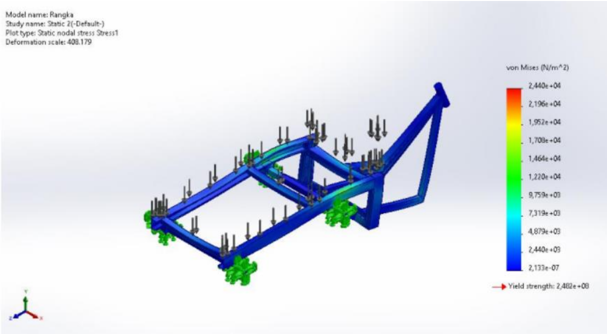

Figure 3. Von-Mises Stress Simulation Results

Stress analysis is an integral part of engineering science, as most failures in engineering components are caused by stress. Stress is defined as the result of forces that cause internal particle reactions and affect an object in different ways, which is important to know when analyzing or designing any structure [17]. Von-Mises stress is the determining factor whether a material will fail or not. The material was considered to start melting when the von-Mises stress reaches a critical value called yield. The Von Misses stress criteria is one of the widely used criteria for designing engineering components of ductile materials. To assess whether a design is within the design limits and will perform safely during its life time design, the Von Mises Stress Criteria proves to be very effective. Von Mises failure theory suggests that a material will fail if the Von Mises stress or effective stress of the material under load is equal to or greater than the yield limit of the same material.

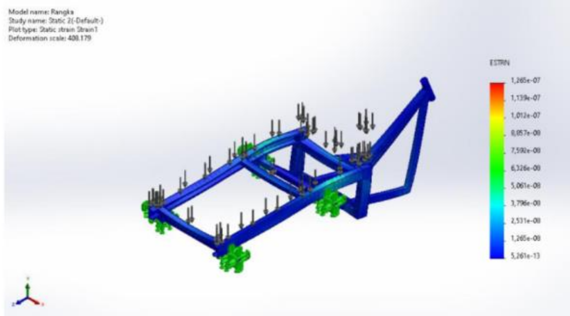

Figure 4. Von-Mises Strain Simulation Result

FEA provides higher accuracy in strain analysis than conventional analysis. Strain analysis is observed based on the mechanical properties of a material and its ability to withstand anticipated loads. Strain hardening is the increase in hardness of a material due to plastic deformation. It is a property that occurs mostly in metals and their alloys, but not all these materials are vulnerable to strain hardening. Strain is the ratio between the deformation and the original length. Think of strain as percent elongation, how much larger (or smaller) an object becomes under load. Figure 4 shows the results of the strain distribution test. The minimum strain produced by simulation testing using SolidWorks is 5.26×10^{-13} and the maximum strain is 1.27×10^{-7}.

Finite Element Analysis (FEA), also called finite element method (FEM), is a numerical method that utilizes displacement. Analysis on elastic bodies is used to develop displacement approximate solutions. The FEM displacement formulation is used to calculate component displacement, strain, and stress analysis. Displacement is required in many precise designs. These results can be used in the design process of solid body mechanics.

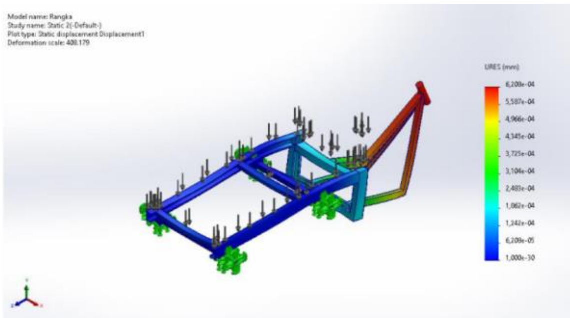

Figure 5. Displacement Simulation Results

The simulation results of static loading on the chassis frame against displacement are shown in Figure 5. The maximum displacement obtained was

6.208×10-4 mm which is marked in red. The location of the maximum displacement is in the steering area. The displacement value indicates that, when the chassis frame is given a load of 1500 N, the chassis frame would have displacement of only 6.208×10-4 mm.

CONCLUSION

Static loading analysis of the chassis frame of a three-wheeled electric vehicle using SolidWorks software has been successfully carried out. The test results show that the maximum stress that occurs in the frame due to the load acting on the chassis is 0.0244 MPa. This value is still far below the maximum stress required by the material, which indicates the safety factor of the chassis frame that has been designed is quite high. The displacement of the frame due to the load produces a low value, which is 6.208×10-4. Based on the simulation results, it can be concluded that the chassis frame design made for three-wheeled electric vehicles can withstand loads up to 1500 N.

SUGGESTION

Testing with other materials is required, so that comparative data will be obtained.

Acknowledgment

For the completion of this journal, thanks to the Directorate General of Vocational Education, Jember State Polytechnic Jember and PT. Manufactur Dynamic Indonesia for providing funding and facilities and assistance through Matching Fund activities in 2023.

REFERENCES

[1] Li, W., Stanula, P., Egede, P., Kara, S. and Herrmann, C., 2016. Determining the main factors influencing the energy consumption of electric vehicles in the usage phase. *Procedia Cirp,* Vol. 48 pp.352-357.

[2] Bae, C., Kim J., 2017. Alternative fuels for internal combustion engines. *Proceedings of the Combustion Institute*, Vol. 36 (3) pp. 3389–3413.

[3] Kalghatgi, G., 2018. Is it really the end of internal combustion engines and petroleum in transport?. *Applied energy*, Vol. 225 pp. 965-974.

[4] Ary, A.K., Prabowo, A.R. and Imaduddin, F., 2020. Structural assessment of an energy-efficient urban vehicle chassis using finite element analysis–A case study. *Procedia Structural Integrity*, Vol. 27 pp. 69-76.

[5] Aziz, M., Oda, T. and Kashiwagi, T., 2015. Extended utilization of electric vehicles and their re-used batteries to support the building energy management system. *Energy Procedia*, Vol. 75 pp. 1938-1943.

[6] Oda, T., Aziz, M., Mitani, T., Watanabe, Y. and Kashiwagi, T., 2017. Actual congestion and effect of charger addition in the quick charger station: Case study based on the records of expressway. *Electrical Engineering in Japan*, Vol. 198 (2) pp. 11-18.

[7] Lagnelöv, O., Larsson, G., Larsolle, A. and Hansson, P.A., 2023. Impact of lowered vehicle weight of electric autonomous tractors in a systems perspective. *Smart Agricultural Technology*, Vol. 4 pp. 100156.

[8] Tubiello, F.N., Salvatore, M., Ferrara, A.F., House, J., Federici, S., Rossi, S., Biancalani, R., Condor Golec, R.D., Jacobs, H., Flammini, A. and Prosperi, P., 2015. The contribution of agriculture, forestry and other land use activities to global warming, 1990–2012. *Global change biology*, Vol. 21 (7) pp. 2655-2660.

[9] Intergovernmental Panel on Climate Change, 2015. Agriculture, Forestry and Other Land Use (AFOLU). *Climate Change 2014: Mitigation of Climate Change*, pp.811-922.

[10] Nandhakumar, S., Seenivasan, S., Saalih, A.M. and Saifudheen, M., 2021. Weight optimization and structural analysis of an electric bus chassis frame. *Materials Today: Proceedings*, Vol. 37 pp. 1824-1827.

[11] Waseem, M., Ahmad, M., Parveen, A. and Suhaib, M., 2022. Inertial relief technique based analysis of the three-wheeler E-vehicle chassis. *Materials Today: Proceedings*, Vol. 49 pp. 354-358.

[12] Guron, B.R., Bhope, D.V. and Yenarkar, Y.L., 2013. Finite element analysis of cross member bracket of truck chassis. *IOSR Journal of Engineering*, Vol. 3 (3) pp. 10-16.

[13] Ghazaly, N.M., 2014. Applications of finite element stress analysis of heavy truck chassis: survey and recent development. *Journal of Mechanical Design and Vibration,* Vol. 2 (3) pp. 69-73.

[14] Hadimani, S., Kumar, S. and Sridhar, S., 2018. Modelling and structural analysis of two-wheeler chassis frame. *International Journal of Engineering Development and Research*, Vol. 6 (1) pp. 907-913.

[15] Patil, R.V., Lande, P.R., Reddy, Y.P. and Sahasrabudhe, A.V., 2017. Optimization of three wheeler chassis by linear static analysis. *Materials Today: Proceedings*, Vol. 4 (8) pp. 8806-8815.

[16] Vijayakumar, M.D., Kannan, C.R., Manivannan, S., Vairamuthu, J., Tilahun, S. and Ram, P.B., 2020, December. Finite element analysis of automotive truck chassis. *In IOP Conference Series: Materials Science and Engineering*, Vol. 988 (1) pp. 012114.

[17] Widiyanto, I., Sutimin, S., Laksono, F.B. and Prabowo, A.R., 2021. Structural assessment of monocoque frame construction using finite element analysis: A study case on a designed vehicle chassis referring to ford GT40. *Procedia Structural Integrity*, Vol. 33 pp. 27-34.