International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 6, Issue 11, pp 64-69, November-2022

https://doi.org/10.47001/IRJIET/2022.611008

Structural Analysis of an Electric Utility Vehicle (EUV) Chassis Using Finite Element Analysis

[1]*Dodeye Ina Igbong, [2]Raymond Rowland Ana, [3]Oku Ekpenyong Nyong, [4]Oliver Ibor Inah, [5]Emmanuel Coco-bassey

[1,3,4,5]Department of Mechanical Engineering, Faculty of Engineering, University of Cross River State, Calabar, Nigeria

[2]Department of Mechanical Engineering, Faculty of Engineering, University of Calabar, Nigeria

*Corresponding Author's E-mail: dodeyeigbong@gmail.com

***Abstract* - Chassis is the main weight support structure of the electric utility vehicle (EUV). It serves an important function in determining the best safety level and size of the construction. Therefore, it's precise design and the use of finite element analysis to simulate the strength of the Chassis and know how much load it can sustain, is imperative. This study aims to design a Chassis that is strong enough and can support the loads and operations of an electric utility vehicle for University of Cross River State (UNICROSS) campus security patrol and emergency response. The work commences with detailed modelling of Chassis using Autodesk Inventor Professional 2019 software and then finite element simulation using a static structural feature in ANSYS 19 software. Results of simulation obtained reveal that equivalent Stress, equivalent elastic strain, total deformation and safety factor are 268.59Mpa, 4.5039e-011 mm/mm, 13.639mm and 15, respectively.**

Keywords: EUV Chassis; Stress and Strain; Deformation; Safety Factor; Finite Element Simulation; ANSYS.

I. INTRODUCTION

Electric Utility Vehicles (EUV) are environmentally sustainable technology that uses electric motors and batteries to produce propulsion. With the current drive to reduce the emission of greenhouse gases such as CO_2 from combustion of fossil fuels, EUVs are attracting attention as potential technology option for eliminating vehicular emission. EUVs are commonly powered by on-board batteries, and have wide applications not limited to transportation of payloads/persons within airports, university campuses, subway stations, sporting arenas, hospitals and any closed area where lifting cargoes quietly is required. To design an EUV to efficiently meet the load and operations for university (UNICROSS) campus security patrol and emergency response, the vehicle chassis design must be precise to provide sufficient mechanical performance such that it does not experience any plastic deformation during loading of any form [1]. It should also be able to satisfy the criteria of lower weight for the same power requirement, which will automatically result in increased efficiency and more fuel savings[2].

The chassis is the main support structure of the vehicle that holds together all the internal components such as the steering system, engine, suspension system, power transmission system, brake system, wheels, passenger and other fittings[3].There are four types of chassis commonly used in the vehicle manufacturing industry, which are monocoque, space frame, chassis backbone, and ladder frame[4], [5]. Monocoque is a type of chassis where the chassis structure is fused together in the vehicle body. A rather popular chassis is the ladder chassis also known as body-on-frame chassis. As the name suggest, ladder frame have shape that resembles a ladder and are favourite for their ability to sustain large loads. However, because of their critical functionality the design of chassis, load analysis and potential failure modes must be identified and corrected through finite element simulation. In engineering, Finite Element Analysis (FEA) is being used to gain better understanding of the material rigidity, toughness, behavior under static and dynamic loads or vibrations throughout its life cycle. Therefore, this paper aims to determine the EUV chassis strength and safety level to support the loads and operations of the EUV for campus security patrol and emergency responses, using AUTODESK INVENTOR PROFESSIONAL 2019 and ANSYS FLUENT 19.

1.1 Chassis

Chassis serves as the main support of a vehicle that keeps the car rigid, stiff and not bending. It is usually made of steel material that holds other components of the car together [6]. The material must have power to hold or support the load on the vehicle[3]. The ladder chassis frame consists of four longitudinal rails which are connected by a series of shorter members called as cross-members, which are attached to these longitudinal rails to provide strength and prevent twisting of the frame. This is a simple design and can be used for multiple applications with slight modifications. Fig.1 shows the EUV chassis model that is designed using Autodesk Inventor Professional 2019 software.

International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 6, Issue 11, pp 64-69, November-2022

https://doi.org/10.47001/IRJIET/2022.611008

There are several studies on vehicle chassis design and analysis.[7]Studied the finite element simulation of an electric-car chassis. In that work, the torsional stiffness and maximum von mises stress on the chassis due to the application of twist load and three other type of loadings (vertical, lateral and braking) were determined.[8]Conducted the numerical modelling of the frame structure of a light van-type electric truck. Stress, strain and modal analyses were performed using ABAQUS software to determine frame design safety and rationality.

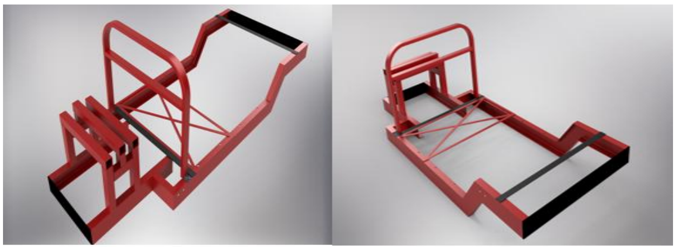


Figure 1: Design of an electric car chassis using Autodesk Inventor professional 2019

In another study, [9] modelled the frame geometry and conducted finite element analysis to determine the chassis structure performance based on material type, material thickness and defined loads.[10]Investigated the impact of material type on the structural performance of a two-wheeler chassis frame using ANSYS software. From static and modal analyses, the authors concluded that Carbon epoxy material showed better performance compared to other materials considered. The vehicle chassis up till today continues to experience development and improvements. Additional studies on vehicle chassis frame that might be of interest is found in [11]–[15].

1.2 Finite Element Analysis Method

The finite element method is a numerical method used in solving engineering and mathematical problems of a physical phenomenon. Types of technical and mathematical physical problems that can be solved by elemental methods are stress / stress analysis, Buckling and Vibration Analysis[3].

- Less cost and calculation time

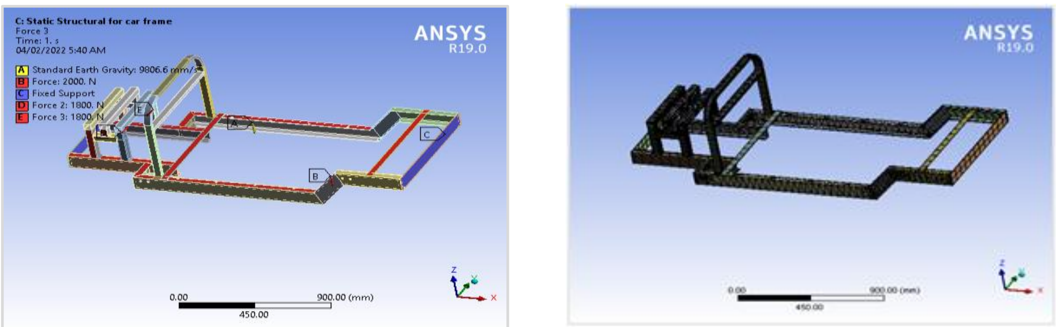
FEA and ANSYS reduces cost and save time during component/system development and design stage. The researcher is not required to physically build the chassis model to examine what happens when loads are applied on the system. In which case it would be even harder to determine certain mechanical properties, for example; to count the moments and determine shear stress at dynamic conditions. Imagine the struggle to move the chassis at the given velocity. That is why it is effective to reduce cost and it will require more effort, more time, and the researcher would want to use his time wisely so it will results in a productive analysis. That is why using finite element method and ANSYS as a tools can save plenty of time [16].

II. METHODOLOGY

The EUV chassis or frame CAD model was first developed in AUTODESK INVENTOR PROFESSIONAL 2019 software, with detail sizing and dimensions to meet design requirements. The CAD model is then simulated and FEA perform on individual components to investigate the stress, strain, deformation, and safety factor of the frame when subjected to different loads and operations. The material selected for this EUV chassis structure is mild steel and its technical specification is presented in Table 1.

2.1 Simulation

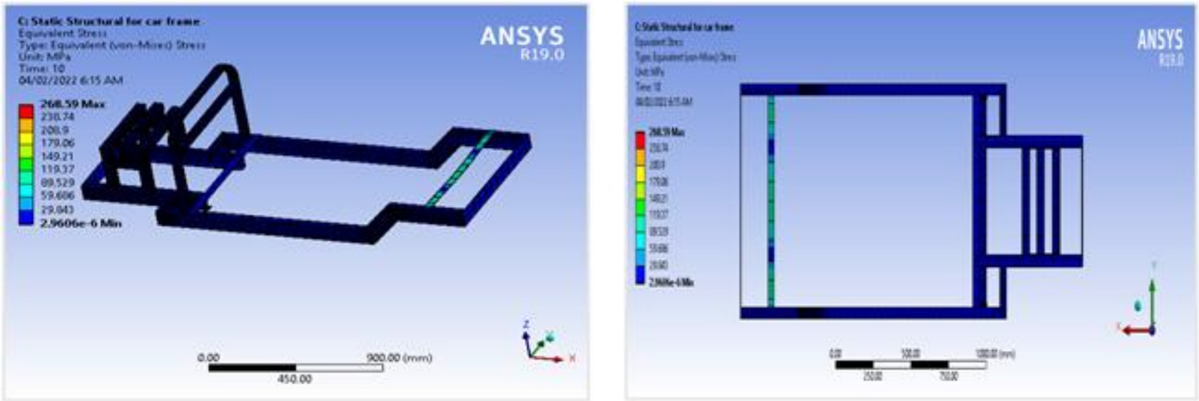
The developed model of the chassis structure is then analyzed using ANSYS Workbench 19 to determine the level of stress, deformation, and safety factors associated with the frame when subjected to loads. This process generated a mesh with 63615 nodes and 16431 elements. In this simulation, the chassis is assigned forces at certain points which represent the loads subjected on the frame with a total of 5600N styles. Fig. 2 illustrates the chassis geometry simulated with mild steel material.



(a) **(b)**

Figure 2: Static structural analysis result (a) and meshing result (b)

ISSN (online): 2581-3048
Volume 6, Issue 11, pp 64-69, November-2022
https://doi.org/10.47001/IRJIET/2022.611008



(a) (b)

Figure 3: Maximum equivalent stress of the EUV Chassis

Table 1: Material Properties

Name	Steel, Mild	
General	Mass Density	7.58 g/cm^3
	Yield Strength	207 MPa
	Ultimate Tensile Strength	345 MPa
Stress	Young's Modulus	220 GPa
	Poisson's Ratio	0.275 ul
	Shear Modulus	86.2745

III. RESULT AND DISCUSSION

3.1 Equivalent (Von-Mises) Stress

Results of finite element simulation are shown in Fig. 3(a)-(b), with minimum and maximum stress on the chassis indicated as 2.9606-6MPa and 268.59MPa, respectively. The value of the minimum stress indicates the lowest stress due to force that can be applied to the structure before deformation starts occurring. Similarly, the maximum stress refers to the highest stress the structure can sustain before complete deformation sets in.

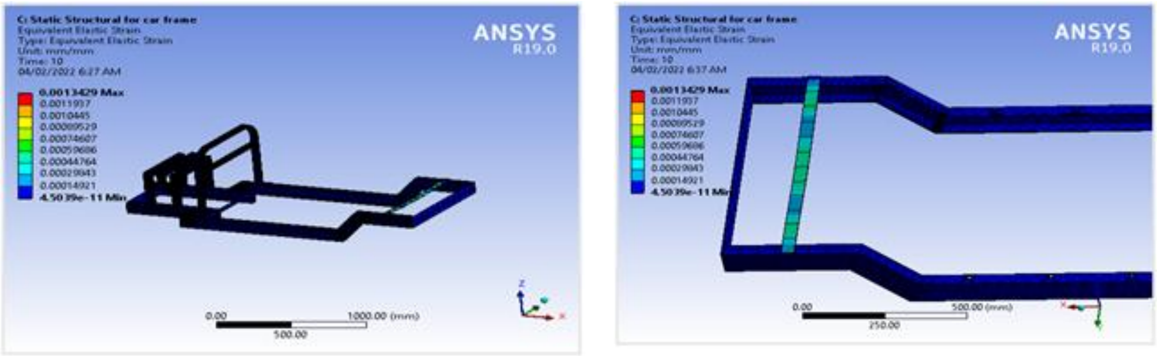
3.2 Equivalent Elastic Strain

In Fig. 4(a) – (b), the simulation results of the equivalent elastic strain is presented. The red colour mark indicates the maximum equivalent elastic strain of 1.3429e-003 mm/mm, and the blue portion represent the minimum equivalent elastic strain of 4.5039e-011 mm/mm.

3.3 Total Deformation

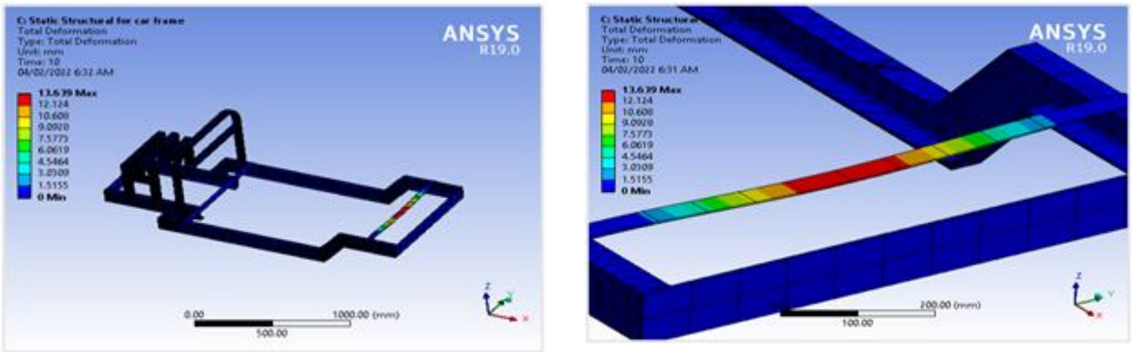
From the analysis it was discovered that when the chassis is subjected to the setup conditions, there is a minimum and maximum deformation of 0 mm and 13.639mm, respectively, as shown in Fig. 5(a)-(b). The total deformation that occurs when the material fails completely is the maximum deformation. At this point, the component is fully distorted and cannot function properly.

ISSN (online): 2581-3048
Volume 6, Issue 11, pp 64-69, November-2022
https://doi.org/10.47001/IRJIET/2022.611008



(a) (b)

Figure 4: Maximum equivalent elastic strain of the EUV Chassis



(a) (b)

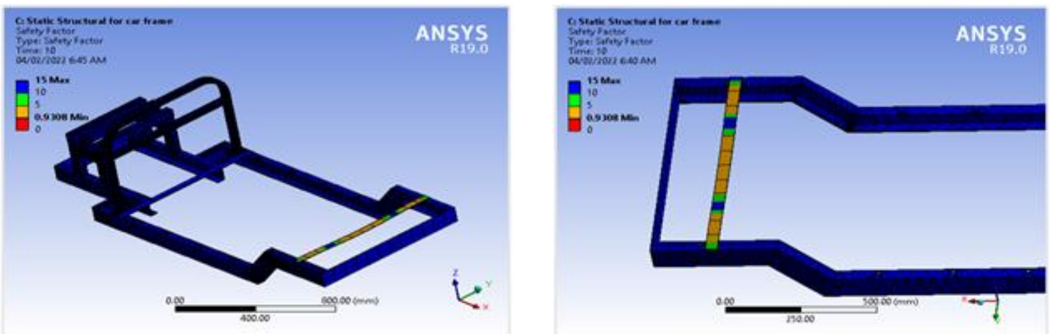
Figure 5: Total Deformation of the EUV Chassis

3.4 Safety Factor

Fig. 6(a)-(b) illustrates the factor of safety of the frame, with the minimum and maximum values obtained as 3.0807 and 15, respectively. The minimum values indicates that for highly stressed areas, failure will occur at a load 3 times the applied pressure, while the maximum value indicate that for areas with less stress concentration, the structure will fail at load 15 times the applied load.

3.5 Fatigue

Fig. 7(a)-(b) illustrates the fatigue life of the EUV frame, estimated minimum life of 3524102.8 months and a maximum life of 380517500 months. The minimum fatigue life predicts how long the chassis will last before failure of any sort starts to occur, beyond which the structure no longer functions effectively. On the other hand, the maximum fatigue life shows how long the material will last before complete failure of the structure occurs.



(a) (b)

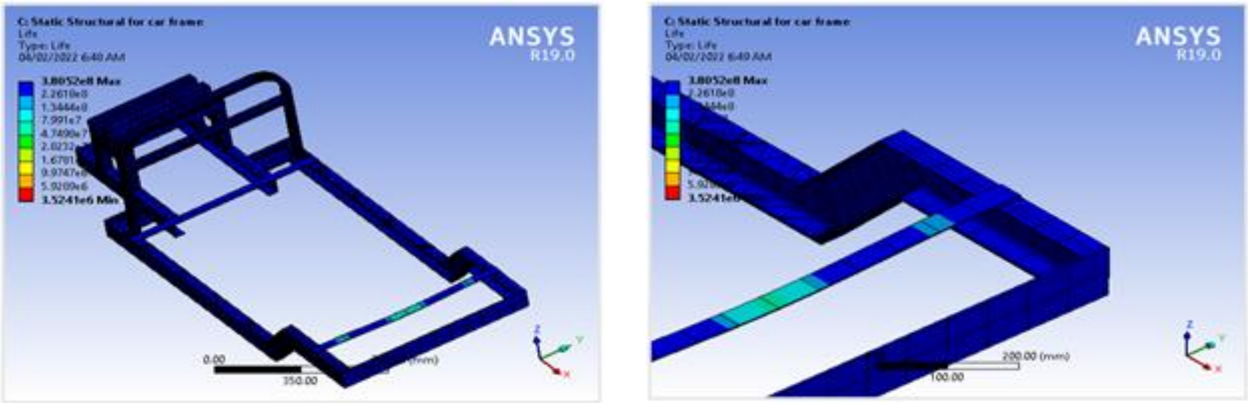
Figure 6: Safety Factor of the EUV chassis

International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 6, Issue 11, pp 64-69, November-2022

https://doi.org/10.47001/IRJIET/2022.611008



(a) (b)

Figure 7: Fatigue life of the EUV Chassis

IV. CONCLUSION

From the calculation and simulation using ANSYS Workbench 19 software, the conclusion of the EUV chassis analysis with mild steel material structure is presented in Table 2.

Table 2: Conclusion

	Total Deformation (mm)	Equivalent Stress (MPa)	Equivalent Strain (mm/mm)	Factor Of Safety	Fatigue life. (Months)
Minimum	0	2.9606-6	4.5039e-011 mm/mm	0.9308	3524102.8
Maximum	13.639mm	268.59	1.3429e-003 mm/mm	15	380517500
Average	0.16042mm	3.6147	2.091e-005 mm/mm		

ACKNOWLEDGEMENT

Thanks to the Federal Government of Nigeria through the office of the Tertiary Education Trust Fund (TETFUND) for sponsoring this research through the Institutional Based Research (IBR) grant. A final thanks to the Department of Mechanical Engineering, University of Cross River State where the research was conducted.

REFERENCES

[1] A.Airale, M. Carello, and A. Scattina, “Carbon fiber monocoque for a hydrogen prototype for low consumption challenge,” *Materwiss. Werksttech.*, vol. 42, no. 5, pp. 386–392, 2011.

[2] A.Kaluza, S. Kleemann, F. Broch, C. Herrmann, and T. Vietor, “Analyzing decision-making in automotive design towards life cycle engineering for hybrid lightweight components,” *Procedia CIRP*, vol. 50, pp. 825–830, 2016, doi: 10.1016/j.procir.2016.05.029.

[3] D. Prasetiyo, Strenght analysis chassis of UM electric cars using finite element method, vol. 07017. 2018.

[4] A.K. Ary, A. R. Prabowo, and F. Imaduddin, “Structural Assessment of an Energy-Efficient Urban Vehicle Chassis using Finite Element Analysis – A Case Study,” *in Procedia Structural Integrity*, 2020, vol. 27, no. 2019, pp. 69–76, doi: 10.1016/j.prostr.2020.07.010.

[5] H. B. P. Hemant B.Patil, “Stress Analysis of Automotive Chassis with Various Thicknesses,” *IOSR J. Mech. Civ. Eng.*, vol. 6, no. 1, pp. 44–49, 2013, doi: 10.9790/1684-0614449.

[6] A.C. Raghuvanshi, T. Srivastav, and R. K. Mishra, “Design and Development of Foldable Kart Chassis,” *in 4th International Conference on Materials Proceedings and Characterization*, 2015, vol. 2, pp. 1707–1713, doi: 10.1016/j.matpr.2015.07.004.

[7] A.Wikarta, R. Hakim, A. Shiddieqy, and I. Sidharta, “Finite Element Simulation for Electric-Car Chassis Development from Scratch,” *Math. Model. Eng. Probl.*, vol. 8, no. 2, pp. 279–284, 2021.

ISSN (online): 2581-3048
Volume 6, Issue 11, pp 64-69, November-2022
https://doi.org/10.47001/IRJIET/2022.611008

[8] H. Zhang, G. Huang, and D. Yu, "Numerical modeling for the frame structure of light van-type electric truck," *Sci. Prog.*, vol. 103, no. 2, pp. 1–27, 2020, doi: 10.1177/0036850420927204.

[9] J. Wang, C. Shi, N. Yang, H. Sun, Y. Liu, and B. Song, "Strength , Stiffness , and Panel Peeling Strength of Carbon Fiber-Reinforced Composite Sandwich Structures with Aluminum Honeycomb Cores for vehicle body," *Compos. Struct.*, 2017, doi: 10.1016/j.compstruct.2017.10.038.

[10] S. Hadimani and S. Sridhar, "Modelling And Structural Analysis Of Two Wheeler Chasis Frame," *Int. J. Eng. Dev. Res.*, vol. 6, no. 1, pp. 907–913, 2018.

[11] A.Chugh, R. Ahuja, and S. Ranjan, "Shape Optimization of Automobile Chassis," *Int. J. Eng. Res. Technol.*, vol. 6, no. 02, pp. 644–648, 2017.

[12] J. C. Brown, A. J. Robertson, and S. T. Serpento, Motor Vehicle Structures: Concepts and Fundamentals. *Elsevier Ltd*, 2001.

[13] A.Gauchia, V. Diaz, M. J. Boada, and B. . Boada, "Torsional stiffness and weight optimization of a real bus structure," Torsional stiffness Weight Optim. *a real bus Struct.*, vol. 11, no. 1, pp. 41–47, 2010.

[14] L. L. Thompson, S. Raju, and E. H. Law, "Design of a Winston Cup Chassis for Torsional Stiffness," *SAE Trans.*, vol. 10, no. 6, pp. 2571–258, 1998.

[15] V. Parlaktras and E. Tanik, "Design of a verylight L7e electric vehicle prototype," *Int. J. Automot. Technol.*, vol. 6, 2015.

[16] N. A. Sutisna, M. F. Aulia, and A. Akbar, FEM Simulation of Electric Car Chassis Design with Torsional Bar Technology, vol. 3, no. 2. 2018.

Citation of this Article:

Dodeye Ina Igbong, Raymond Rowland Ana, Oku Ekpenyong Nyong, Oliver Ibor Inah, Emmanuel Coco-bassey, "Structural Analysis of an Electric Utility Vehicle (EUV) Chassis Using Finite Element Analysis" Published in *International Research Journal of Innovations in Engineering and Technology - IRJIET*, Volume 6, Issue 11, pp 64-69, November 2022. Article DOI https://doi.org/10.47001/IRJIET/2022.611008

****

