Ocean Engineering : Jurnal Ilmu Teknik dan Teknologi Maritim
Vol. 2 No. 3 September 2023
e-ISSN: 2963-5454, p-ISSN: 2963-5012, Hal 143-157
DOI: https://doi.org/10.58192/ocean.v2i3.1158

Designing Of The Ngalah Data Electric Car Frame Using The Finite Element Method

Rifky Zaidani
University of Yudharta Pasuruan

Mochamad Mas'ud
University of Yudharta Pasuruan

Jl. Yudharta No.7, Kembangkuning, Sengonagun, Kec. Purwosari, Pasuruan, Jawa Timur, Indonesia
Email : zaidanirifky98@gmail.com

***Abstract**. The development of fossil fuel transportation continues to increase massively every time, and this condition is terrible news for the environment worldwide because the impact of gas emissions produced from burning fossil fuels is quite damaging to the environment. For this reason, based on the problems above, every country, including Indonesia, needs to contribute to maintaining and caring for environmental conditions from the threat of damage caused by pollution from burning fossil fuels. One of the efforts made by the Indonesian government to overcome this is to develop electric vehicles in collaboration with* KEMRISTEK *through the Indonesian Electric Car Championship (*KMLI*) event, which involves universities in* Indonesia*. One of the main parts of an electric car is the frame or chassis; for this reason, the author tries to design the best Frame for an electric car. Other leading factors such as the value of Stress, Deformation, and Factor of Safety, the method researchers use in this study is the Finite Element Method. The results showed that the frame did not fail and could meet the safety standards that the author had set.*

***Keywords**: Chassis; Deformation; Factor of Safety; Finite Element Method; Standards.*

Abstrak. Perkembangan transportasi berbahan bakar fosil terus mengalami peningkatan secara masif setiap waktunya, kondisi ini merupakan kabar buruk bagi lingkungan diseluruh dunia dikarenakan dampak dari gas emisi yang dihasilkan dari proses pembakaran bahan bakar fosil cukup merusak bagi lingkungan. Untuk itu berdasarkan permasalahan diatas, setiap negara termasuk Indonesia perlu berkontribusi dalam menjaga dan merawat kondisi lingkungan dari ancaman kerusakan yang diakibatkan dari pencemaran pembakaran bahan bakar fosil. Salah satu upaya yang dilakukan pemerintah indonesia dalam mengatasi hal ini adalah dengan mengembangkan kendaraan listrik yang bekerjasama dengan KEMRISTEK lewat ajang kejuaraan mobil listrik indondesia (KMLI) yang melibatkan berbagai Universitas di Indonesia. Salah satu bagian utama dalam mobil listrik adalah rangka atau sasis, untuk itu penulis mencoba merancang sebuah sasis terbaik untuk mobil listrik Ngalah Data Universitas Yudharta Pasuruan berdasarkan pada standard regulasi KMLI dan penelitian dari mahasiswa ITENAS untuk standard kemanan rangka, selain itu penelitian ini juga memperhatikan beberapa faktor utama lainnya seperti nilai Tegangan, Deformasi, dan *Factor of Safety*, metode yang peneliti gunakan dalam penelitian kali ini adalah Metode Elemen Hingga. Hasil penelitian menunjukkan rangka tidak mengalami kegagalan dan mampu memenuhi standard kemanan yang telah penulis tentukan.

Kata kunci: Deformasi, *Factor of Safety*, Metode Elemen Hingga, Sasis, Standard.

Received Mei 30, 2023; Revised Juni 30, 2023; Accepted Juli 20, 2023
* Rifky Zaidani, zaidanirifky98@gmail.com

INTRODUCTION

To support the continuity of their life, humans will not be separated by technological developments, especially in transportation, be it land, sea, or air. As a supporter of human mobility, conventional or non-renewable energy-fueled transportation continues to experience a relatively high increase in the number of products. Even though it has positive impacts, such as lower prices and increasingly sophisticated integrated technology, on the other hand, this has also caused various negative impacts, such as the impact of environmental damage due to the massive amount of exhaust gas released into the air. Therefore, every country must be committed to working together to overcome the above problems (Setiyana, 2021).

One effort that can be realized is to gradually reduce the number of conventional vehicles and the need to increase the production of electric vehicles or Electric Vehicles (EVs) on a large scale because, in general, the characteristics of EVs do not emit exhaust gases into the environment (Budiman, 2021).

The Indonesian government, through the Ministry of Research Technology, and Higher Education (KEMRISTEKDIKTI), is holding a competition called KMLI, or the Indonesian electric car competition, this championship challenges every university to develop their electric car, and by holding this championship, it is hoped that it will produce reliable engineers in producing electric cars by the nation's young generation (Imbang, 2019).

One part that is quite a challenge in developing a vehicle is designing the frame or chassis because this is the main component of a vehicle where each sub-component rests on this Frame, such as motors, gearboxes, drive seats, etc. So that various types of stress can occur here, be it static or dynamic (Tyagi, 2016).

In prioritizing the safety side, more is needed to have material parameters regarding strength or rigidity. Toughness and hardness are also crucial aspects that a design engineer needs to pay attention to in designing an object that risks a person's safety. (Chen, 2018).

To answer the above problems, developing and testing a Frame design can be carried out using scientific computational methods where each process is run by special software. This method is called the Finite Element Method, with the most widespread use in engineering. By depending on various factors such as the accuracy of data input and models, this method allows the accuracy of various test results such as the value of Stress, Deformation, and Factor of Safety close to real conditions in the field. (William, 2021).

Standardization is a detailed list of technical requirements and the process of developing and implementing specifications based on the agreed views of companies, users, interest groups, and governments. The resulting standards are intended to prioritize suitability, safety, and quality. Standards can be developed and regulated by a Standards Development Organization or independently, for example, by a company with a first mover or dominant position in the market. (Xie, 2016).

THEORETICAL STUDY

Ansys assists (Durgam, 2021) in validating the strain, stress, and strain stress results from the strain gauge when analyzing the strength of the frame structure of the electric vehicle made by their team. When subjected to a load of 400kg at a certain point, these observations show that the frame strength meets the required criteria.

(Meti, 2021) with the help of Solidworks software, designed a city car-type electric car frame using a ladder-type frame with a C-section cross-section with dimensions of 100x130x3 mm. Based on static simulation tests via Solidworks, the ITENAS Bandung student design did not fail as long as the Stress value was still far below the permitted limit for the material used, that is 46.96 Mpa. In contrast, the allowable stress value for the material was around 275Mpa, and the maximum deflection was 2.096 mm, and for Factor of Safety by 6.

Through the Static Analysis feature, the Inventor software makes it easier for (Hidayat, 2017) to test the strength of the shell eco marathon vehicle frame for the urban concept type. This study can be concluded that the frame design is still safe because the primary stress value is 183 MPa or still below the material's maximum allowable stress value. Then, for Deformation, it is still relatively small at 3.35 mm.

(Mohammed, 2018) found the expected life and weak points of the formula vehicle Frame using the Finite Element Method, which is run by Ansys software. Through its dynamic analysis feature, the simulation is carried out by applying vibrational forces to engine parts to find out how the Frame behaves during the simulation process running. This observation obtained a vibration value of 155 Hz after the natural frequency simulation was carried out on the analysis modal. For static, maximum stress of 235 MPa and 1.3 mm for maximum deflection, all results are packaged in a graphic that is easy for readers to understand.

Based on the various studies and literature sources above, the author will design and test the design strength of the KMLI Ngalah Data car frame using Solidworks software to create 3D design modeling and use Ansys Software to perform Finite Element Method analysis.

This research was conducted to produce an electric car frame design and find the most significant stress value from the 3D design of the Ngalah Data Electric Car frame when given a certain amount of loading. In Figure 1, you can see the design form of the framework that the author will examine.

Fig. 1, Frame Model

Table 1. Frame Weight and Length

No	Type	Mass (Kg)	Length (cm)
1	Pipe Hollow 25x25x1.9	9.3	2114
2	Square hollow 20x20x1.8	3.4	436
3	Pipe Hollow 25x25x2.3 (Rollbar)	1.4	140
4	**Total**	**14.1**	**2690**

METODE PENELITIAN

Currently, the Finite Element method is a method of solving structural, fluid, and cross-disciplinary problems in physics which is very popular in engineering; this method can solve very complicated mathematical problems where complex structures are divided into several parts. -small parts which are pieces, and these tiny parts are referred to as Finite Elements; these elements are created by a process called meshing; it should be noted that the results of solving problems through this method give the results of stresses, deformations, and the like in the form of estimates only (Maski, 2018).

One technique that can be used is matrix operations. However, the selection of this technique is limited to certain types of elements, one of which is a cubic element involving 24 degrees of freedom because it has 8 points where each point has 3 degrees of freedom. In the finite element simulation process, each part can consist of hundreds of thousands to millions of elements, and even in some instances, the number can be far above that, so completing this method with manual calculations is almost impossible (Sutisna, 2021).

Analysis using this method can easily predict various physical phenomena that will occur after the analysis process is carried out by displaying specific values as validation parameters. An example of its application in the automotive world is when an engineer wants to analyze the strength of a Frame. This method allows product development to be carried out at a lower cost because physical prototyping, considered to require a large amount of money, can be eliminated (Bathe, 2008).

For details of Finite Elements, it can be seen in Figure 2. Where the areas in the Tetrahedral and Quadrilateral sections are Finite Elements, black dots are called (nodes) that connect one element to another. At the same time, the mesh (mesh) is a unified structure of nodes and elements.

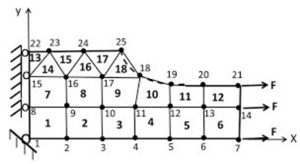


Fig. 2, Finite Element Configuration

1. FRAME MATERIAL

In this study, researchers assume that the Ngalah Data electric car frame uses Aluminum 6061-T6. Researchers consider this material the most appropriate choice because Aluminum is a light metal so it will reduce the vehicle's weight. This material can help make the vehicle easier to control by providing better braking and steering levels. Aluminum also absorbs vibrations better than steel, so it will help protect the driver from harm in the event of an accident. The following is the cross-sectional profile of 25x25x1.9/2.3 Hollow Pipe and 20x20x1.8 Square Hollow, along with the specifications for Aluminum 6061-T6 properties.

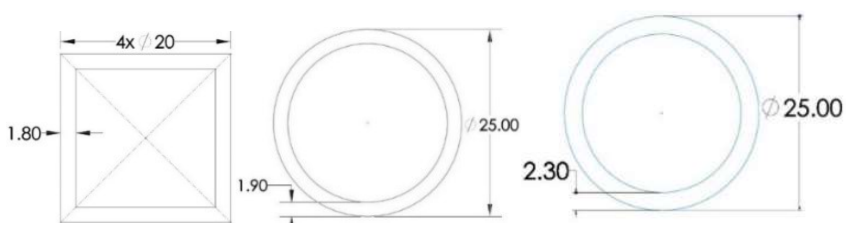
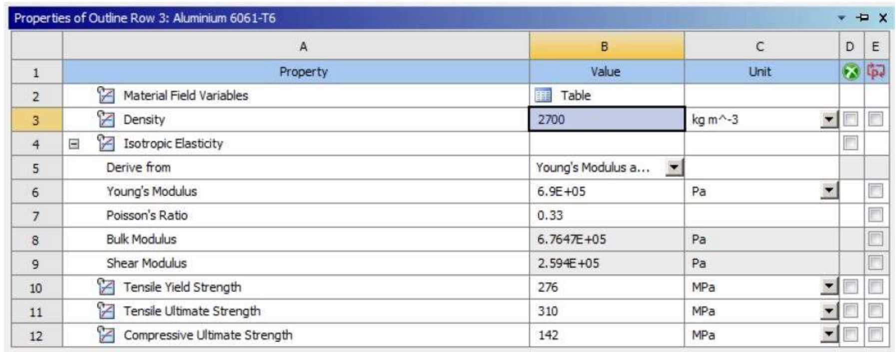


Fig. 3, Material Cross Section Profile



Properties of Outline Row 3: Aluminium 6061-T6

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	2700	kg m^-3		
4	Isotropic Elasticity				
5	Derive from	Young's Modulus a...			
6	Young's Modulus	6.9E+05	Pa		
7	Poisson's Ratio	0.33			
8	Bulk Modulus	6.7647E+05	Pa		
9	Shear Modulus	2.594E+05	Pa		
10	Tensile Yield Strength	276	MPa		
11	Tensile Ultimate Strength	310	MPa		
12	Compressive Ultimate Strength	142	MPa		

Fig. 4, Aluminium 6061-T6 Specification Property

2. DEFINING THE TECHNICAL SPECIFICATIONS

Specifications provide clear instructions about project intent, performance, and construction. This section may refer to the quality and standards that must be applied. Manufacturing materials and products can be clearly defined.

In this section, the author determines the technical specifications of the material and the design criteria and frame safety standardization based on projects from ITENAS Bandung students (Meti, 2021), which was carried out last November 2021, and the KMLI regulations, as for the following technical specifications.

a). Criteria Design

Table 2. standardization targets

No	standardization	Criteria
1	Geometri	It fits 1 Driver within the required dimensions.
2	Material	Best *Yield strength* not > 270 Mpa
		Light
3	*Safety*	Max Deformation 5 mm
		Factor of Safety >1.5
		The stress does not exceed 75% of the Yield Strength of the material.

Sumber: (Meti, 2021)

b). Frame Dimension

The next standardization target that researchers need to achieve is the dimensions of the Frame itself which must not violate the rules set by the organizers (KMLI). In KMLI's guide document regarding dimensions, conditions are only specified for rollbar height and the vehicle's width. We adjust the rest, such as wheelbase, vehicle length, etc., based on reference forms from projects on the Internet. Following are the target dimensions of the KMLI Ngalah Data Car.

Table 3. Frame Dimension Target

Dimension		
NO	Section	Parameter (mm)
1	Width+Wheels	1200 - 1400
2	Rollbar height	≥ 50 from driver helmet

Sumber: (Imbang, 2019)

c). Fixed and Load Support

Support in this design is located at the bottom of the frame, or more precisely, around the rear and front axle. In this section, the author uses the fixed support feature to determine the position of the support frame. For the location of the load on the frame design, the author assumes the driver's position is supine but tends to sit. Many other universities apply for this ideal position in designing electric car frames that will be included in the KMLI competition. The following can be seen in Figure 5, the position in question.

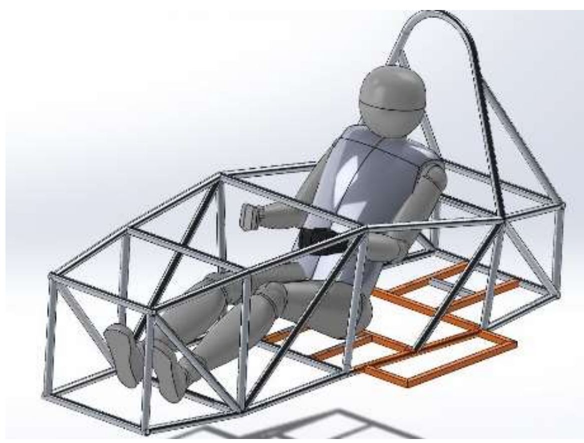


Fig. 5, Illustration of Driver Sitting Position

Based on Figure 5. the following is the load distribution that will be simulated via the Ansys software.

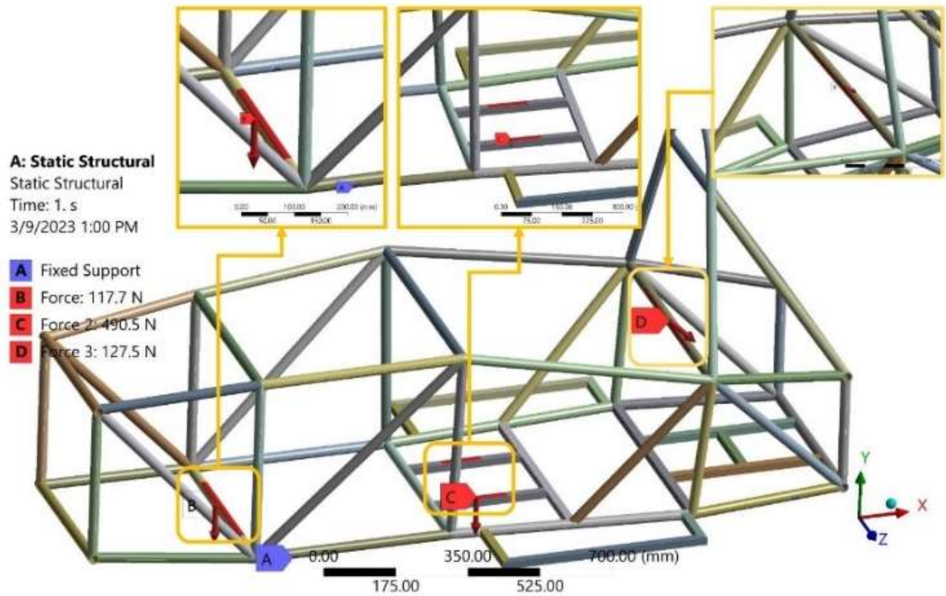


Fig. 6, Point of Load in Detail

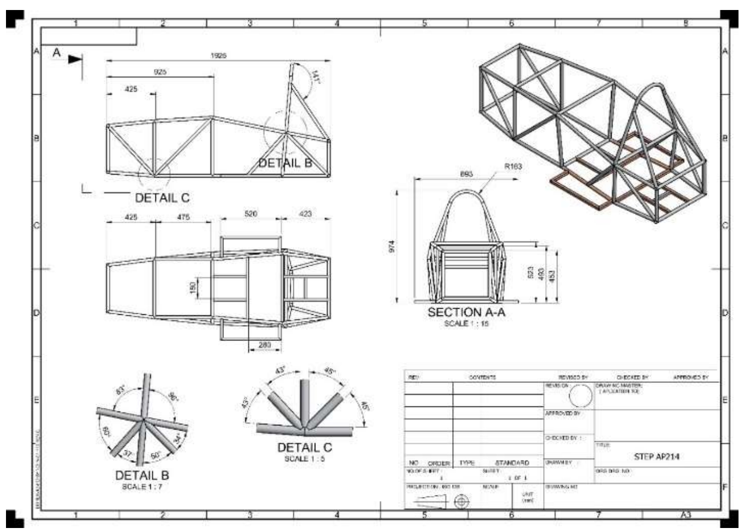


Fig. 7, Dimension of the Frame Model

d). Finite Element Method Simulation

The next step in this research process is the simulation of the Finite Element Method through the static structural features contained in the Ansys software. To maintain and ensure the accuracy of the results, various parameters must be considered in carrying out the simulation either through Ansys software or another finite element simulation-based software, one of which is producing a mesh or breaking objects into small parts such as nets.

Producing a mesh that meets the criteria is a foundation in engineering simulation science because computers solve problems based on formulas and various physical and mathematical equations through mesh shapes. Therefore, the more regular, uniform, and smaller the resulting mesh, the computer will accurately process available data.

The following is the average recommended quality of the mesh spectrum according to Ansys experts based on the Skewness and Orthogonal Quality mesh criteria (Adam, 2020).

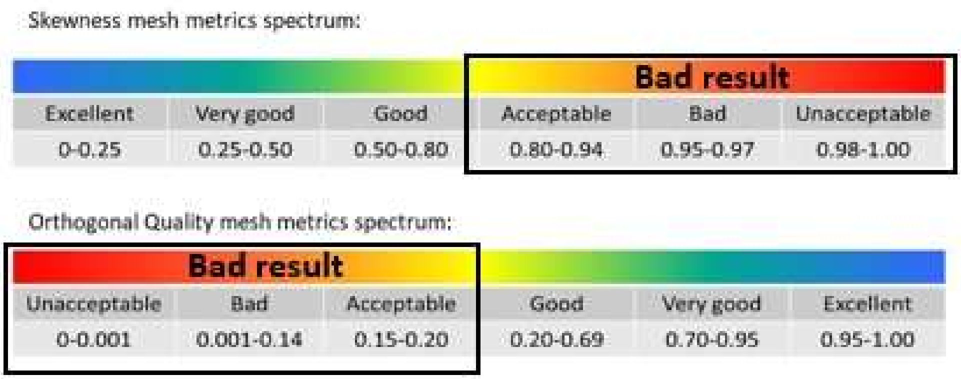


Fig. 8, Mesh Quality Recomendation

To achieve optimal mesh results and meet the mesh quality recommendations above, at this stage, the author uses a mesh size of (6.5 mm) and chooses the Tetrahedron mesh type for all parts of the frame. Uniformity of mesh types like this is also an excellent effort to improve mesh quality. Besides that, the author also uses the refinement feature to produce smaller and more detailed meshes in certain parts. The mesh results can be seen in Figure 9 and Table 4 for the mesh criteria data

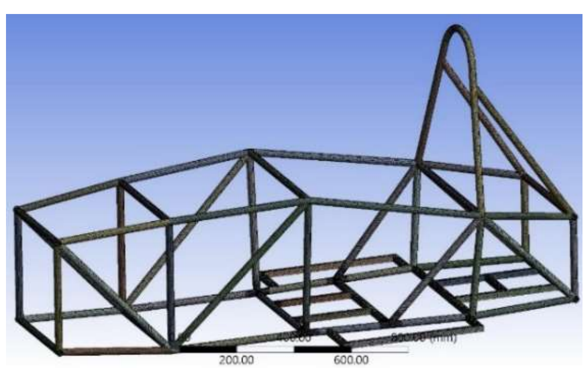


Fig. 9, Meshing Result and Refinement Point

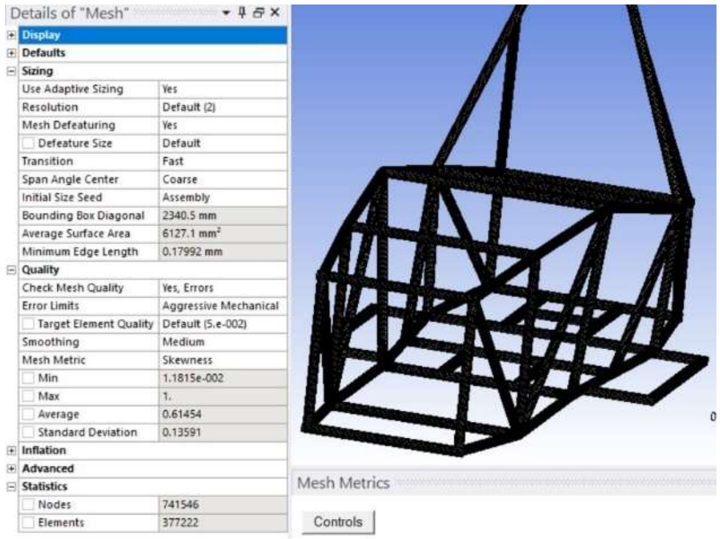


Fig. 10, Detail of Mesh Spesification

Table 5 shows the data from the mesh results. The data obtained for the average skewness value is 0.61; for the resulting elements, there are more than 377 thousand elements. As shown in Figure 8 regarding the recommendations for mesh quality, the results of the author's re-mesh are in the "good" range, or it can be concluded that the mesh construction process was successfully carried out. The 3D frame model can be simulated in the next stage.

Table 4, Mesh Charateristic

No	Characteristic		Value
1	Avg. Element Size (% of model size)		6.5 mm
2	Span angle center		Coarse
4	Avg Surface Area		6.127 mm²
6	Nodes		741.546
7	Element		377.222
8	Skewness	Min.	1.1815e-002
		Max.	1
		Avg.	0.61

RESULT AND DISCUSSION

This section will discuss the results of the Static Structural simulation of the Ngalah Data Electric car frame via the Ansys software after it is assumed to receive a load distribution of

75 kg. This process aims to produce output in the form of Von Misses Maximum Voltage values, Factor of Safety, and Deformation.

The result of the Von Misses Maximum Stress value, as shown in Figure 11, is 91.1 Mpa and is located on the frame support rod, which is connected to the back of the car frame or, more precisely, on the right side of the driver's waist. This value means that the stress that occurs is still below the standard limit that the author has set in the table 2. Besides that, based on the Von Mises stress theory, which is stated as follows, the material can still be safe if the equivalent stress or Von Mises stress is still below the allowable stress value. The frame does not experience significant failure after being subjected to a force of 75 kg**.**

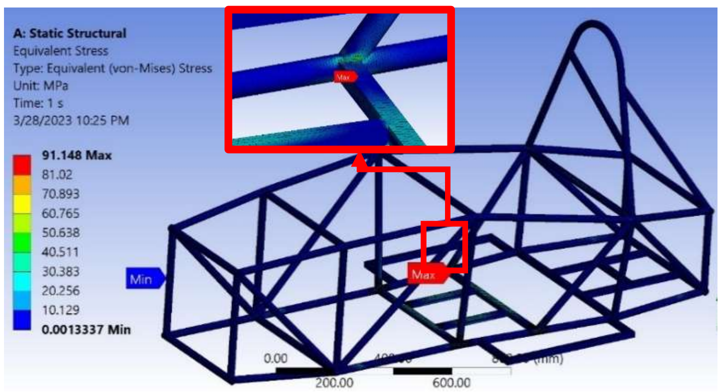


Fig. 11, Max Stress

Then Figure 12 shows a deformation of 0.12 mm, located just below where the majority load is concentrated, that is on the driver's central support rod, located at the bottom of the wheelhouse. The deformation value is still below the predetermined standard limit of 0.5mm, Table 2.

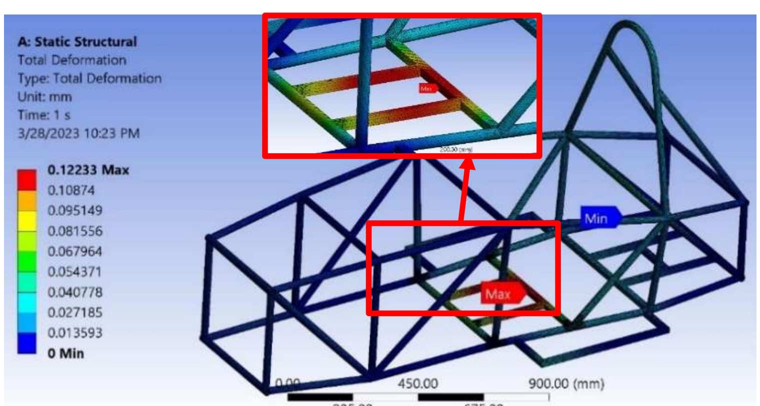


Fig. 12, Deformation Value

Another Finite Element Method analysis result produced by Ansys software is the Factor of Safety, which is one of the parameters in determining the safety of a design based on the applied stress value; this value is generated from the ratio of the allowable stress divided by the stress that occurs on an object when subjected to a force with a specific value.

As shown in Figure 13, the value of the Factor of Safety for the Ngalah Data electric car frame is 2.65, and it is located at the bottom of the primary connection to the right of the driver support; that means that this value meets the Factor of Safety standards set in table 2, which is > 1.5, then the design can be said to be safe for static loading. In contrast, the conditions will undoubtedly differ in field application, so dynamic loading must also be considered. Therefore, the safety of the frame design based on the results of the Factor of Safety value is further strengthened by the definition of the safety factor value, which states that the ideal safety factor value for objects that will be subjected to a dynamic force is ranged 2-3 (Dobrovolsky, 1978).

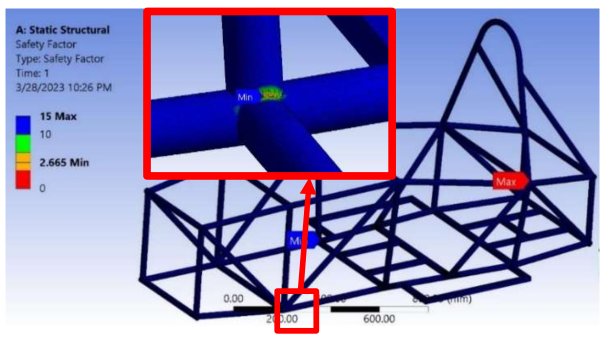


Fig. 13, Factor of Safety Value

Figures 11 to 13 show the results of the Finite Element Method simulation of the Ngalah Data electric car frame based on the mechanical properties of Aluminum 6061-T6 material with a particular load distribution.

The simulation is carried out to determine various parameters in determining the safety of a design, such as the value of Stress, Deformation, and Factor of Safety. Referring to the data from the simulation results, which show that the corresponding parameter values are still below the permissible limits of the material used, namely Aluminum 6061-T6, it can be concluded that the frame did not experience significant failures during the simulation process.

Table 5. Frame Strength Standard Inspection

No	standardization	Criteria	Frame Design
2	Material	Best Yield strength not > 270 Mpa	Aluminium, 276 MPa
		Light	2700 kg/m^3
3	Safety	Max Deformation 5 mm	0.12 mm

		Factor of Safety >1.5	2.65
		The stress does not exceed 75% of the Yield Strength of the material.	91.1 Mpa

Table 6. Frame Dimension Standard Inspection

Dimension			
NO	Section	Parameter (mm)	Frame Design (mm)
1	Width+Wheels	1200 - 1400	1925
2	Rollbar height	≥ 50 from driver helmet	±85

CONCLUSION

Here are some conclusions that the authors draw based on the results of the above research:

a. After testing the strength through the Ansys Workbench Software simulation, the maximum deformation value occurs at 0.12 mm for a stress value of 91.1 MPa or below 75% of the yield strength limit for Aluminum 6061-T6 material. For a safety factor value of 2.65, all of these values still meet the standard targets set by the authors referring to previous research, so it can be said in terms of simulation testing, the Ngalah Data electric car frame meets the strength standards from previous studies and will not experience significant failure when subjected to a static load distribution of 75Kg at points certain point.

b. From the technical drawings provided, in terms of the dimensions of the Ngalah Data electric car frame, it meets the regulatory criteria or standards from the organizers of the KMLI.

c. Large companies' engineers widely use simulation testing through supercomputer sophistication to save costs. However, physical trials through product prototypes also need to be carried out, considering various limitations that computers cannot duplicate with actual conditions.

Suggestion

In order to obtain more optimum test results, here are some topics that also need to be studied in further research:

d. Prediction of the degree of torsional stiffness and its effect on vehicle behavior.

e. Lateral and longitudinal force simulations are crucial to study more deeply to identify new critical stress points.

b. The need to involve other elements in terms of loading, such as components that have a dominant load.

BIBLIOGRAPHY

Setiyana, B. (2021). Analisis dan Modifikasi Rangka Mobil Antawirya Menggunakan Metode Elemen Hingga. ROTASI. 23(2), 50-56. https://doi.org/10.14710/rotasi.23.2.%p.

Budiman, Febrian Arif, et al. (2021). Analisis Tegangan Von Mises dan Safety Factor pada Chassis Kendaraan Listrik Tipe In-Wheel. Jurnal Rekayasa Mesin. Vol.16, No.1, hal. 100-108.

Imbang Rachmat (2019). Kompetisi Mobil Listrik Indonesia Xi - Tahun 2019. Bandung: Politeknik Negeri Bandung.

Tyagi, Apoorva. (2016). Design and Analysis of a Space Frame Tubular Chassis for a Formula Student Car. International Journal of Innovative Research in Science, Engineering and Technology. Vol. 5, Issue 9.

Chen, S.-J., Chen, C.-Y., & Lin, M.-R. (2018). Risk factors for crash involvement in older motorcycle riders. Accident Analysis & Prevention, 111, 109–114. doi:10.1016/j.aap.2017.11.006

William, Al. (2021). Finite Element Analysis Vs Real World: Hackaday. https://hackaday.com/2021/05/27/finite-element-analysis-vs-real-world/. Dikunjungi tanggal 14 Maret 2023.

Xie, Z., et al (2016). Standardization efforts: The relationship between knowledge dimensions, search processes and innovation outcomes. Technovation, 48-49, 69–78. doi:10.1016/j.technovation.2015.12.002.

Durgam, Shankar. et al (2021). Experimental and Numerical Studies on Materials for Electric Vehicle Chassis. IOP Conf. Series: Materials Science and Engineering. doi:10.1088/1757-899X/1126/1/012073

Meti, Y. D., Nugraha, M. P., Marsono (2021). Analisis Statik Chassis Mobil Listrik Jenis Ladder Frame Dengan Batang Struktur Honeycomb Berbahan Alumunium Alloy Dengan Bantuan Software Solidworks. Rekayasa dan Aplikasi Tekik Mesin di Industri. 43-51.

Hidayat, Taufik. et al (2017). Perancangan dan Analisis Statik Chassis Kendaraan Shell Eco Marathon tipe Urban Concept. Jom FTEKNIK. Volume 4 No. 2.

Mohammed, N. A., Nandu, N. C., Krishnan, A., Nair, A. R., & Sreedharan, P. (2018). Design, Analysis, Fabrication and Testing of a Formula Car Chassis. Materials Today: Proceedings, 5(11), 24944–24953. doi:10.1016/j.matpr.2018.10.295.

Maski, S., & Basavaraj, Y. (2015). Finite element analysis of engine mounting bracket by considering pretension effect and service load. International Journal of Research in Engineering and Technology (IJERT), 4(08), 327-333.

Sutisna, N. A. (2023). Design Analysis of a Tubular Chassis for an Electric Vehicle using Finite Element Method. Jurnal Teknik Mesin dan Mekatronika (Journal of Mechanical Engineering and Mechatronics), 8(1), 37-51.

Bathe, K.-J. (2008). Finite Element Method. Wiley Encyclopedia of Computer Science and Engineering. doi:10.1002/9780470050118.ecse159.

Adam, Nor & Attia, Osam & Al-Sulttani, Ali & Mahmood, Hussein & As'arry, Azizan & Anas, Khairil & Md Rezali, Khairil Anas. (2020). CFD Letters Numerical Analysis for Solar Panel Subjected with an External Force to Overcome Adhesive Force in Desert Areas. CFD Letters. 12. 60-75. 10.37934/cfdl.12.9.6075.

Dobrovolsky, K. Z. V., Machine elements: a textbook. Moscow: Peace Publisher, 1978.