Jurnal Asiimetrik: Jurnal Ilmiah Rekayasa Dan Inovasi
Accredited "**SINTA 3**" based on the Decree of Dirjendiktiristek, Number: 225/E/KPT/2022
Volume 7 Number 2, July 2025, Page 267-278
Website : http://journal.univpancasila.ac.id/index.php/asiimetrik/

p-ISSN 2655-1861
e-ISSN 2716-2923

Analysis of the Construction Form of an Electrical Car Chassis Prototype Type Hollow Alumunium 6061 Pipe Profile Using Inventor Software 2019

Analisis Bentuk Konstruksi Chassis Mobil Listrik Prototipe Tipe Hollow Alumunium 6061 Profil Pipa Menggunakan Software Inventor 2019

Aep Surahto[1], Riyan Ariyansah[2], Fathoni[1], Qommaruddin[1], Agus Fikri[3*], Mohammad Mujirudin[4], Arry Avorizano[5], Goodman Octavianus[6]
[1]Teknik Mesin, Fakultas Teknik, Universitas Islam 45 Bekasi, Jawa Barat, Indonesia
[2]Teknik Mesin, Fakultas Teknologi Industri dan Informatika, Universitas Muhammadiyah Prof. Dr. HAMKA, Jakarta, Indonesia
[3]Teknik Mekatronika, Fakultas Teknologi Industri dan Informatika, Universitas Muhammadiyah Prof. Dr. HAMKA, Jakarta, Indonesia
[4]Teknik Elektro, Fakultas Teknologi Industri dan Informatika, Universitas Muhammadiyah Prof. Dr. HAMKA, Jakarta, Indonesia
[5]Teknik Informatika, Fakultas Teknologi Industri dan Informatika, Universitas Muhammadiyah Prof. Dr. HAMKA, Jakarta, Indonesia
[6]Teknik Mesin, Fakultas Teknik dan Ilmu Komputer, Universitas Global Jakarta, Jawa Barat, Indonesia.

Article information:

Received:
16/06/2025
Revised:
28/06/2025
Accepted:
05/07/2025

Abstract

The chassis is a critical structural component in vehicle design, especially in electric vehicle (EV) development aimed at reducing environmental impact. This study analyzes the structural performance of a prototype electric vehicle chassis made from 6061 hollow aluminum using Autodesk Inventor 2019. Three chassis models were designed and tested under a static load of 900 N to compare stress distribution, displacement, and safety factors. All chassis designs had dimensions of 2160 mm × 750 mm × 500 mm with a wall thickness of 2 mm. Simulation results showed that the H-type chassis model performed best, with a von Mises stress of 36.03 MPa, displacement of 4.28 mm, and a safety factor of 7.63. These results indicate that the H-type chassis is structurally safe and suitable for prototype application.

Keywords: prototype, inventor, von mises stress, displacement, safety factor.

***SDGs*:**

Abstrak

Chassis merupakan komponen struktural yang sangat penting dalam desain kendaraan, khususnya dalam pengembangan kendaraan listrik (EV) yang bertujuan mengurangi dampak lingkungan. Penelitian ini menganalisis performa struktural chassis prototipe mobil listrik berbahan aluminium hollow 6061 menggunakan perangkat lunak Autodesk Inventor 2019. Tiga model chassis dirancang dan diuji secara statik dengan pembebanan sebesar 900 N untuk membandingkan distribusi tegangan, deformasi, dan faktor keamanan. Seluruh model memiliki dimensi 2160 mm × 750 mm × 500 mm dengan ketebalan 2 mm. Hasil simulasi menunjukkan bahwa model chassis tipe H memiliki kinerja terbaik dengan tegangan von Mises sebesar 36,03 MPa, deformasi 4,28 mm, dan faktor keamanan sebesar 7,63. Berdasarkan hasil tersebut, chassis tipe H dinyatakan aman dan layak digunakan sebagai chassis prototipe.

Kata Kunci: prototipe, inventor, tegangan von mises, displacement, faktor keamanan.

*Correspondence Author
email : agus_fikri@uhamka.ac.id

1. INTRODUCTION

The chassis is the most important structural part of the vehicle (Rao *et al.*, 2017). Nowadays, many vehicles use electric power, because fossil fuels will run out. The existence of electric fuel makes today's vehicle environmentally friendly and does not emit pollutions. The development of the era has made the automotive world now participate in the development that used to be chassis made of steel then developed using steel plates and even aluminum which have a light weight so that they get the material (Hirsch, 2011; Tisza and Czinege, 2018).

Electrical car technology is now also a solution as an energy saver that is now running low in the world (Farghali *et al.*, 2023). There is also transportation that can use two version of energy by using electricity and gasoline which work alternately when the vehicle is in a particular vehicle, so that fuel use is more efficient. The chassis is a supporting component in supporting the load given to a car (Mat and Ghani, 2012; Kumar and Deepanjali, 2016). The chassis generally has several types that are widely known, such as the ladder frame chassis, this type of chassis is used in this study, aluminum space frame, and monocoque. The chassis in this study is expected to be able to support the load from the engine to the driver.

In the development of electric vehicles, a lightweight yet strong chassis structure is essential to enhance energy efficiency and ensure safety. Computer-Aided Engineering (CAE) technology offers an effective solution for digitally designing and analyzing structural components (Lee and Han, 2009). Using CAE software such as Autodesk Inventor, engineers and researchers can model complex chassis structures and simulate static loads to predict material behavior without physical prototypes (Ellianto and Nurcahyo, 2020; Zamzam *et al.*, 2025). This approach enables detailed evaluation of stress distribution, deformation, and safety factors, leading to more efficient and reliable designs (Wibowo *et al.*, 2022; Hendarmin, Hanifi and Naubnome, 2023). This study applies such a method to analyze the structural performance of a prototype electric vehicle chassis made from hollow aluminium 6061 pipe.

The Finite Element Method (FEM) is a widely used numerical approach for analyzing structural strength in vehicle chassis design, particularly for electric vehicles (Tsirogiannis, Stavroulakis and Makridis, 2019; Ary, Prabowo and Imaduddin, 2020; Durgam *et al.*, 2021). According to Abbas et al., FEM enables designers to simulate stress and deformation distribution across a chassis structure before fabrication, improving design efficiency and safety (Abbas, Juma and Jahuddin, 2020). In their study, Autodesk Inventor was used for modeling, while Autodesk Robot Structural Analysis was employed for strength evaluation. The chassis, made of AISI 1020 steel in a tubular space frame configuration inspired by the BMW i3, showed a maximum stress of 35.443 MPa, maximum strain of 0.00450474, and maximum displacement of 1.3164 mm. The results confirmed that the chassis design was able to withstand the given loads without failure (Abbas, Juma and Jahuddin, 2020). These findings serve as a reference for the present study, which focuses on analyzing a prototype electric vehicle chassis constructed from aluminum 6061 hollow pipe using Autodesk Inventor 2019.

Hendrawan et al., designed the chassis electric vehicle prototype to achieve an optimal balance between structural strength and dimensional efficiency (Hendrawan *et al.*, 2018). Their study employed a ladder frame configuration due to its simplicity and proven rigidity, using Square Tube Aluminium Alloy 6063-T6 as the main material. The chassis was modeled using SolidWorks Premium 2016, and structural analysis was conducted using the software's built-in Finite Element Analysis (FEA) tools. The simulation outputs included Von Mises stress, displacement, and safety factor values. The analysis revealed a maximum Von Mises stress of 2.15×10^7 N/m², a maximum displacement of 1.31 mm, and a safety factor of 2.6, indicating that the structure can withstand the applied loads. Furthermore, rollbar simulations demonstrated its ability to support a load of up to 700 N. These results provide useful benchmarks for the current study, which analyzes an electric vehicle chassis using aluminium 6061 pipe profiles, focusing on structural performance under static loading (Hendrawan *et al.*, 2018).

This study uses a ladder frame type with hollow aluminum material to maximize performance and costs. In this study, Autodesk Inventor 2019 software was used, which includes finite element simulation (FEM) and has a 3-Dimensional (3D) model feature that can solve difficulties in design and analysis science. This ladder frame chassis hopes to find out the distribution pf stress, displacement, and safety factor with a driver load of 60 kg and other load variables. The prototype chassis used must be strong enough to support heavy loads in the form of loads from the engine, accessories, acceleration, and the driver's load itself. A strong chassis requires materials with the appropriate quality.

The strength test must be carried out on the chassis with stress testing, frame testing, and safety testing. This test is carried out using Autodesk Inventor 2019 to measure the capabilities of the chassis to be tested so that the chassis can be used safely for drivers and avoid unwanted things. The chassis is also an important part of every vehicle. In designing this chassis analysis, it is expected to get optimal result at the level of safety of this chassis and the dimensions of the chassis, and the dimensions of the chassis construction are adjusted to be able to withstand the load of all components that will later be placed on the chassis. In the test simulation to obtain accurate results, the testing process uses Autodesk Inventor 2019 software with the use of stress analysis features to obtain stress, displacement, and safety values to determine the deflection that occurs in the prototype chassis during loading testing.

2. METHODOLOGY

The method in study has stages such as literatur review, data collection for reference in making chassis models, modeling ladder frame chassis design with three patterns, simulation and analysis of chassis designs that have been made to obtain the best chassis.

The literatur review aims to obtain numerical results that will be used as a reference for chassis feasibility testing in this study. The literature review aims to collect numerical values that will be used as a reference in testing the feasibility of the frame in this study. The frame chosen is a ladder frame type that has various advantages and good compatibility in designing electric car prototypes. This study applies finite element analysis with the help of Autodesk Inventor 2019 software. Furthermore, data collection is carried out through systematic observation techniques, namely observations that focus on relevant aspects in the study. The author will discuss the analysis focused on three frame patterns, including von Mises stress analysis, displacement, and safety factors using 6061 aluminum hollow material.

The chassis simulation in this study provides a load of 900 N with this load there is a driver load including a driver's leg load of 100 N, a driver's back load of 300 N, a driver's waist load of 200 N, and a front-wheel drive component load of 100 N. with a total load obtained of 900 N.

Creating a chassis design model is done by starting to make a chassis sketch with a predetermined size along with the type of material to be used. After the chassis model design is completed and formed, the design is simulated with software Autodesk Inventor 2019.

The simulation carried out on the chassis construction design aims to obtain detailed and accurate construction strength values and visual results from the simulation. From the simulation results on the chassis design, three values are obtained including the von mises stress, displacement, and safety factor values, where these values are related to the research.

2.1. Applications and Devices in Research

2.1.1. Hardware

The hardware used in this study is an Asus laptop type X441U with RAM 4GB DDR3L specifications, CPU 2.0 GHz, using a windows 10 home 64-bit system, and an intel Core i3- 6006U processor.

2.1.2. Software

CAD software that can facilitate designing with 2D and 3D modeling. Software is also needed to complete analysis work along with simulation of vehicle chassis to get the best results computationally. The software used is Autodesk Inventor 2019 software (see Figure 1). The software used in this chassis research uses Autodesk Inventor 2019 software, to perform static analysis simulations on the chassis prototype model design to obtain the best chassis design results in this study.

Figure 1. Software Autodesk Inventor 2019.

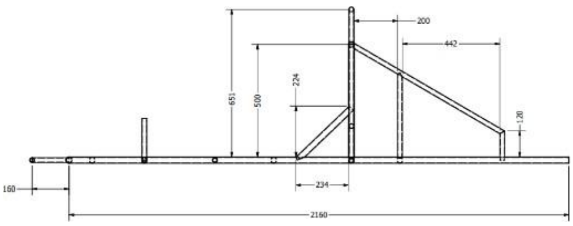
2.2. Chassis Design Model and Sketch

2.2.1. Design and sketch chassis

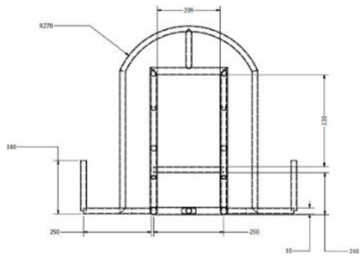
This study created and tested three chassis models, precisely in the middle of the chassis, where this part functions to support the driver. The chassis model in this study uses hollow aluminum 6061 pipe profile material, designed and tested using Autodesk Inventor 2019 software. The chassis in this study has dimensions of 2,160 mm in length, 250 mm in width, and 650 mm in height (see Figure 2a and 2b). The dimensions above are used in the three chassis patterns, which are designed in the middle position of the chassis in the shape of the letters H, X, and Y. can be seen in Figure 2c, 2d, and 2e.

2.2.2. Modeling and simulation

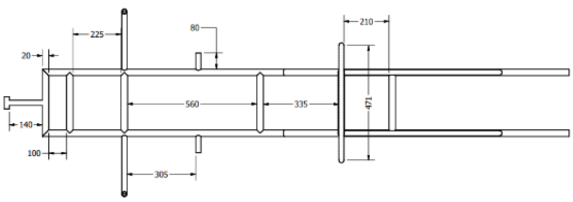
The simulation stage in this study requires appropriate material data, by entering the material into the chassis model to be tested with Autodesk Inventor 2019. Determining several parts of the chassis to determine the part as a



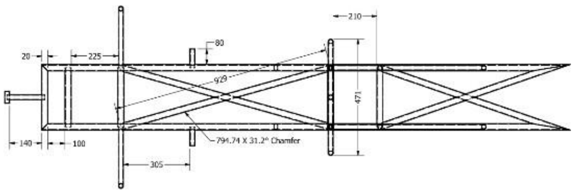
(a) Side view chassis sketch



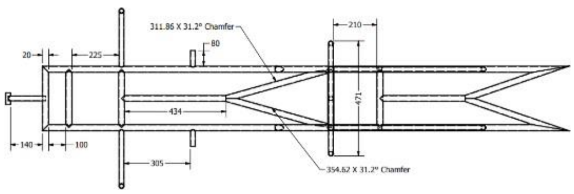
(b) Front view chassis sketch



(c) Sketch chassis H



(d) Sketch chassis X



(e) Sketch chassis Y

Figure 2. Design and sketch chassis.

rigid mount that will be connected to the suspension so that it does not experience displacement due to loading.

The calculation stages of the chassis feasibility test in this study require loading on certain parts as points that will be given loads that correspond to each part of the loading point on the chassis. The loading on the chassis requires a formula to convert kilograms (kg) to Newtons (N), so the calculation is carried out by multiplying the specific gravity by the loading. Loading at predetermined points with adjusted loads is carried out to obtain maximum results from testing on this chassis.

The loading carried out on the three chassis models has the same load and loading point. The loading on the chassis is simulated with Autodesk Inventor 2019 to get the best results from all aspects so that the chassis is suitable for use. The following is the loading carried out on the three chassis patterns with 6061 aluminum hollow pipe profile material.

The following are the loads points given to the chassis of the three chassis models H, X and Y. Loading is given at a predetermined point to withstand the load. The parts that are determined as loading points with their respective loads can be seen in the Figure 3, Figure 4, Figure 5, Figure 6 and Figure 7, with the predetermined load values.

The calculation using equation (1) for the foot load:

$$F1 = \text{foot load} \times g \quad (1)$$

F1 = 10 kg × 1 m/s²
= 100 N

The H, X and Y model chassis design has a load point of 10 kg. The load point is converted by multiplying the gravity value by 10 m/s², so it becomes 100 Newton (N). The load point can be seen in Figure 3.

Here is the calculation using equation (2) for the waist load:

$$F1 = \text{waist load} \times g \quad (2)$$

F1 = 20 kg × 10 m/s²
= 200 N

The H, X and Y model chassis design has a load point of 20 kg. The load point is converted by multiplying the gravity value by 10 m/s², so it becomes 200 Newton (N). The load point can be seen in Figure 4.

The following is the calculation using equation (3) for back load:

$$F1 = \text{back load} \times g \quad (3)$$

F1 = 30 kg × 10 m/s²
= 300 N

The H, X and Y model chassis design has a load point of 30 kg. The load point is converted by multiplying the gravity value by 10 m/s², so it becomes 300 Newton (N). The load point can be seen in Figure 5.

The following is the calculation using equation (4) for engine load:

$$F1 = \text{engine load} \times g \quad (4)$$

F1 = 20 kg × 10 m/s²
= 200 N

The H, X and Y model chassis design has a load point of 20 kg. The load point is converted by multiplying the gravity value by 10 m/s², so it becomes 200 Newton (N). The load point can be seen in Figure 6.

The following is the calculation using equation (5) for wheel drive component:

$$F1 = \text{wheel drive component} \times g. \quad (5)$$

F1 = 10 kg × 10 m/s²
= 100 N

The H, X and Y model chassis design has a load point of 10 kg. The load point is converted by multiplying the gravity value by 10 m/s², so it becomes 100 Newton (N). The load point can be seen in Figure 7.

2.2.3. Analysis static

The next stage, after the design of the three chassis models is given a load at a predetermined point, then perform a static simulation on the three chassis models using 6061 aluminum material. Static simulation in Autodesk Inventor aims to obtain the stress value against the load given to the chassis. At this simulation stage there are colored images ranging from blue to red, these

(a) Chassis H (b) Chassis X (c) Chassis Y

Figure 3. Foot load point.

(a) Chassis H (b) Chassis X (c) Chassis Y

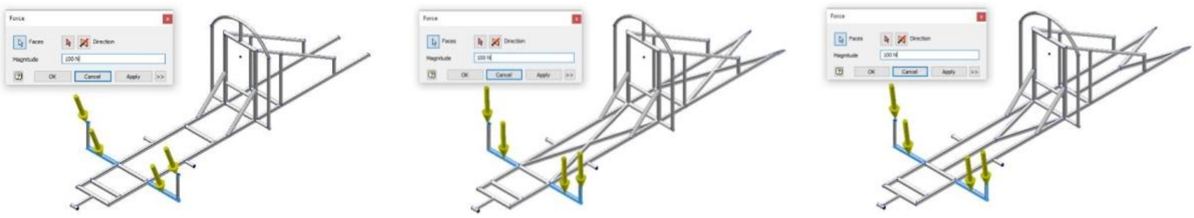
Figure 4. Waist load point.

(a) Chassis H (b) Chassis X (c) Chassis Y

Figure 5. Back load point.

(a) Chassis H (b) Chassis X (c) Chassis Y

Figure 6. Engine load point.



(a) Chassis H (b) Chassis X (c) Chassis Y

Figure 7. Wheel drive component load point.

colors indicate the effect of stress on the loading that produces deflection on the chassis due to the force at the points that are given the load. In this static analysis simulation, the chassis of the three models obtains a stress value, displacement value against the load, and a decent safety value.

Aluminum is a metal element that has a light weight and can conduct electrical energy optimally. The aluminum that will be used is aluminum type 6061. In this study, aluminum material must be considered to obtain maximum results. Each material has its own specifications. The specifications of the aluminum 6061 material obtained from testing in this study as shown in Table 1.

Table 1. Physical properties of aluminum 6061.

Name	Aluminum 6061	
General	Mass Density	2,7 g/cm^3
Stress	Yield strength	275 MPa
	Ultimate Tensile Strength	310 MPa
	Young Modulus	68,9 GPs
	Poisson's Ratio	0,33 μ
	Shear Modulus	25,9023 GPa

2.3. Design Validation

Validation of chassis design with three models in this study is an accurate original design using the Autodesk Inventor 2019 application. Validation as a standard form provides a standard for chassis design to ensure that the design results are feasible or not feasible according to the established standards. The standard obtained as a reference in calculating the test results on the chassis is that the resulting stress value must be below the yield strength value and the safety value must be at a safe level.

2.3.1. Chassis design authentication

Authentication stages in the chassis prototype design process using Autodesk Inventor 2019 software, here are the stages:

1) Preparation for Making Chassis Design
   This process is a preparation of the chassis shape that will be designed with 6061 hollow aluminum material to carry out static testing on loads, according to the needs and desires to get maximum results.
2) Modeling Process
   Chassis design modeling has several stages that aim to facilitate understanding of the chassis model, this design modeling process uses Autodesk Inventor 2019. Here are the stages:
   a. Chassis Material Input
      The design of the chassis frame has an initial stage by making a sketch, after which the sketch is made in 3D form by forming a chassis model according to this research. The three chassis models will be made with ISO standards and dimensions of 21.3 x 2.0 and 26.3 x 2.3. The types of materials used in the design and testing of the chassis are ISO 4019 (Circular Hollow Section - Cold shaped) - Structural steel, Cold - shaped, welding, structural hollow section.
   b. Chassis Finishing Process
      The chassis finishing stage is a process carried out on the chassis rod parts, to form a whole and strong frame in terms of its material structure. Here are the stages:
      (1) Insert Frame
          Entering the type of frame to be used, the standard used in this study uses the ISO standard. The frame used uses a hollow type that can be selected.
      (2) Process Miter
          The miter process is done to help in creating the chassis design, the miter process can be said to include the welding process for the chassis parts, because the software will automatically connect two unrelated components to be connected especially at the corners of the components. The following are the conditions before and after the miter process is carried out.
      (3) Process Trim/Extend
          The process of connecting the ends of components that touch each other by cutting off the excess parts so that they can be connected neatly.

2.3.2. Validation of chassis analysis results

Chassis analysis is carried out as proof in the form of validation in carrying out the analysis stages using software, the analysis is carried out in several stages to obtain accurate and maximum results.

a. Pre-process analysis
   The static analysis procedure on the chassis by providing load points and carrying out the initial stage in the simulation process by verifying the materials used in this study.
   (1) The first stage is to complete the chassis model to carry out the next step, namely conducting a stress

analysis on the three chassis models in this study.

(2) Then in conducting the analysis on Autodesk inventor 2019, you must use the three analysis tools in the environment section and then create a simulation.

(3) Enter the material type in the assign material tool in Autodesk Inventor 2019 with the material type aluminum 6061.

(4) Determine constraint points (support) in chassis design and determine contacts automatically.

(5) Determine the gravity value that corresponds to the angle variable to the chassis.

(6) Input load points with points assigned to each chassis model, with loads assigned to the chassis.

b. Analysis process

Chassis analysis process, after the initial analysis stage has been carried out, the next analysis stage is meshing view, simulation and simulation results, the following are the stages:

(1) Meshing View

The meshing process is used automatically which is useful as a measure of flatness on the frame surface, which makes it easier to analyze frame components with more acurate results.

(2) Simulate

Simulate is the stage of the analysis simulation process that will be run by the software, then click "Run" to start the simulation.

(3) Process Simulation

Waiting for the ongoing simulation process to get a simulation result.

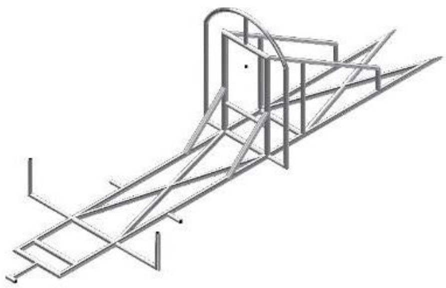
(4) Simulation Result

The simulation results carried out with Autodesk Inventor 2019 in this study will display the stress value with a color graph that shows how much stress value occurs on the chassis against the load.

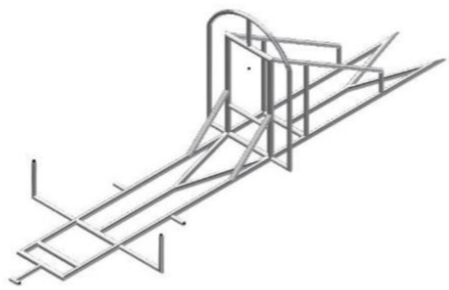
3. RESULTS AND DISCUSSION

Static analysis simulation on the chassis will provide results in the form of values needed in this study, these values are the stress values obtained by the chassis from the loading, the displacement values that occur in the chassis, and the security values to ensure the chassis is at a good level of security. The design H, X and Y model chassis uses 6061 hollow aluminum material. The 3D model of the H, X and Y model chassis as shown in Figure 8.

(a) The H model chassis



(b) The X model chassis



(c) The Y model chassis

Figure 8. 3D model of the H, X and Y chassis.

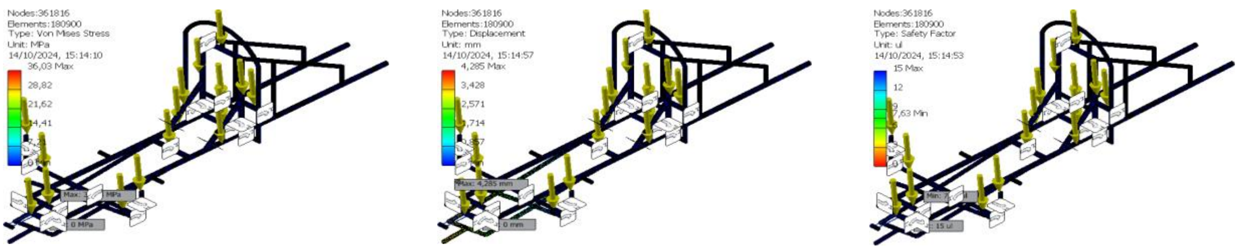
The testing that has been carried out through several stages and methods combined with other methods, then through the help of Autodesk Inventor 2019 software, the test results were obtained from three frame patterns, namely the H, X and Y patterns.

3.1. The H Pattern Chassis

Testing has been done using Autodesk inventor 2019, obtained the stress value obtained by the chassis from the loading, the displacement value that occurs on the chassis, and the safety value to ensure the chassis is at a good safety level. The stress value obtained on the H model chassis is 0 MPa at the minimum value and 36.03 MPa at the maximum value. This frame is included in the safe category because in Figure 9a, the color it has is blue which indicates that there is no excess stress, excess stress occurs if the frame turns red. In this case it can be said to be safe and does not break if given a load of 900 N.

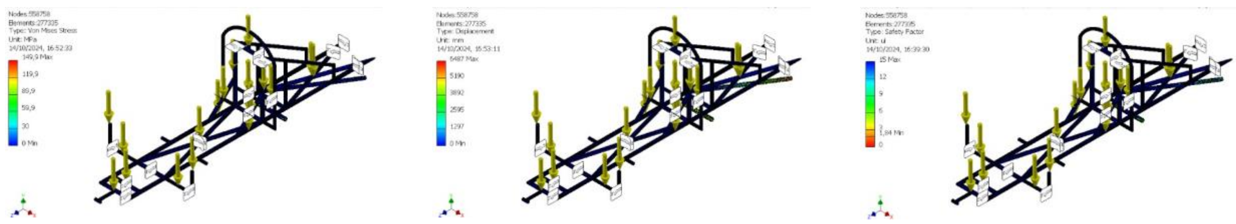
In Figure 9b, it is found that there are several components of varying colors on the chassis, which indicates a high displacement. The displacement value shows a minimum value of 0 and a maximum value of 4.285 mm.

The next test is the safety factor analysis of the H-pattern frame. This frame has a safety factor value of 7.63. It is still considered safe because there is no red color that is close to the lowest value in the frame (see Figure 9c).



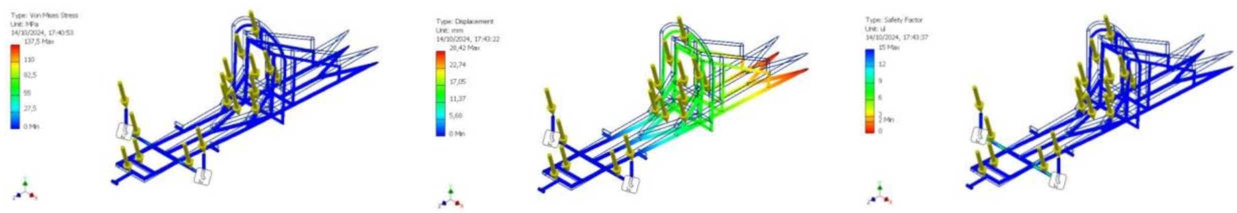
(a) Von Mises stress analysis (b) Displacement analysis (c) Safety factor analysis

Figure 9. The simulation analysis result of the H pattern chassis.



(a) Von Mises stress analysis (b) Displacement analysis (c) Safety factor analysis

Figure 10. The simulation analysis result of the X pattern chassis.



(a) Von Mises stress analysis (b) Displacement analysis (c) Safety factor analysis

Figure 11. The simulation analysis result of the Y pattern chassis.

3.2. The X Pattern Chassis

The displacement value that occurs on the chassis, and the safety value to ensure the chassis is at a good safety level. The stress value obtained on the X models' chassis is 0 MPa at the minimum value and 149.9 MPa at the maximum value. This chassis is included in the safe category because in Figure 10a the color it has is blue which indicates that there is no excess stress, excess stress occurs if the chassis changes color to red. In this case it can be said to be safe and does not break if given a load of 900 N.

The next test is about displacement. In Figure 10b it is found that there are several components of varying colors on the chassis, which indicates a high displacement.

The displacement value shows a minimum value of 0 and a maximum value of 6.487 mm.

The X pattern chassis has a safety factor value of 1.84. The chassis with this X pattern is classified as less safe and cannot be used in high torque applications, because the results are close to the red color which indicates the lowest value in the chassis. The test results can be seen in Figure 10c.

3.3. The Y Pattern Chassis

Testing has been done using Autodesk inventor 2019, obtained the stress value obtained by the chassis from the loading, the displacement value that occurs on the chassis, and the safety value to ensure the chassis is at a good safety level. The stress value obtained on the Y models' chassis is 0 MPa at the minimum value and 137.5 MPa at the maximum value. This chassis is included in the safe category because in Figure 11a the color it has is blue which indicates that there is no excessive stress, excessive stress occurs if the chassis changes color to red. In this case, it is said to be safe, and no fracture occurs if given a load of 900 N.

In Figure 11b it is found that there are several components of varying colors on the frame, which indicates a high displacement. The displacement value shows a minimum value of 0 and a maximum value of 28.42 mm.

The next test is the Y-pattern chassis safety factor analysis. This chassis has a safety factor value of 2. The chassis with this Y pattern is considered less safe and cannot be used in high torque applications, because the results are close to the red color which indicates the lowest value in the chassis (see Figure 11c).

The results of the static analysis simulation on the prototype chassis in this study, which can be seen in the previous images, show the results with the stress values obtained by the chassis from the loading, the displacement values that occur in the chassis, and the safety values to ensure the chassis is at a good safety level obtained from the three chassis models using 6061 hollow aluminum material and the results are summarized in the form of a Table 2.

The results of static analysis using Autodesk Inventor 2019 on the three chassis models in this study, obtained the stress value obtained by the chassis from the loading, the displacement value that occurs in the chassis, and the safety value to ensure the chassis is at a good safety level.

Table 2. Analysis results.

Model	Classification	Results
H Pattern Chassis	Von Mises Stress	36.03 MPa
	Displacement	4.28 mm
	Safety Factor	7.63
X Pattern Chassis	Von Mises Stress	149.9 MPa
	Displacement	6.48 mm
	Safety Factor	1.84
Y pattern Chassis	Von Mises Stress	137.5 MPa
	Displacement	28.42 mm
	Safety Factor	2

The results of the three models obtained different values, the first chassis model with an H pattern shape obtained a stress value (Von Mises Stress) of 36.03 MPa, a displacement value on the chassis of 4.285 mm, and a safety value of 7.63. The second chassis model with an X pattern shape obtained a stress value (Von Mises Stress) of 149.9 MPa, a displacement value on the chassis of 6.487 mm, and a safety value of 1.84. The third chassis model with a Y pattern shape obtained a stress value (Von Mises Stress) of 137.5 MPa, a displacement value on the chassis of 28.42 mm, and a safety value of 2. The difference in the shape of the electric car prototype chassis in this study resulted in different static analysis simulation results.

4. CONCLUSION

Conclusion of prototype chassis construction analysis with three chassis design models using 6061 hollow aluminum material and simulated with the Autodesk Inventor 2019 application, shows the test results on each chassis model with different values, even though the load given to each chassis model is the same. The load given is 900 N on each chassis model, with that the results of the analysis in this study based on the results of the chassis simulation with the H pattern model are the safest chassis and the modeling process is easy in the design process, by obtaining a stress value (Von Mises Stress) of 36.03 MPa, a displacement value on the chassis of 2.28 mm, and a safety value of 7.63 which indicates the chassis at a good safety level. Based on the conclusion on the chassis with the X and Y pattern models, it shows that the chassis with this model still needs further evaluation and improvement to obtain good and appropriate stress values and safety levels.

The research conducted on Autodesk Inventor 2019, with the aim of analyzing three chassis models made of 6061 hollow aluminum which have different models and loads at the same support point, namely 900N, on the three chassis models can be accepted well according to the analysis results

REFERENCES

Abbas, H., Juma, D. and Jahuddin, M.R. (2020) 'Penerapan Metode Elemen Hingga Untuk Desain Dan Analisis Pembebanan Rangka Chassis Mobil Model Tubular Space Frame', *ILTEK : Jurnal Teknologi*, 15(02), pp. 96–102. Available at: https://doi.org/10.47398/iltek.v15i02.32.

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*, 27, pp. 69–76. Available at: https://doi.org/10.1016/j.prostr.2020.07.010.

Durgam, S. *et al.* (2021) 'Experimental and Numerical Studies on Materials for Electric Vehicle Chassis', in *IOP Conference Series: Materials Science and Engineering. 3rd International Conference on trends in Material Science and Inventive Materials (ICTMIM 2021)*, Coimbatore, India: IOP Publishing Ltd, p. 012073. Available at: https://doi.org/10.1088/1757-899X/1126/1/012073.

Ellianto, M.S.D. and Nurcahyo, Y.E. (2020) 'Rancang Bangun dan Simulasi Pembebanan Statik pada Sasis Mobil Hemat Energi Kategori Prototype', *Jurnal Engine: Energi, Manufaktur, dan Material*, 4(2), pp. 53–58. Available at: https://doi.org/10.30588/jeemm.v4i2.753.

Farghali, M. *et al.* (2023) 'Strategies To Save Energy In The Context Of The Energy Crisis: A Review', *Environmental Chemistry Letters*, 21(4), pp. 2003–2039. Available at: https://doi.org/10.1007/s10311-023-01591-5.

Hendarmin, D.W., Hanifi, R. and Naubnome, V. (2023) 'Perancangan Struktur Mobil Listrik "JETZ" dan Analisis Statik Menggunakan Fea (Finite Element Analysis)', *Mutiara: Multidiciplinary Scientifict Journal*, 1(10), pp. 526–531. Available at: https://doi.org/10.57185/mutiara.v1i10.74.

Hendrawan, M.A. *et al.* (2018) 'Perancanganchassis Mobil Listrik Prototype "Ababil" dan Simulasi Pembebanan Statik dengan Menggunakan Solidworks Premium 2016', in *Proceeding of The 7th University Research Colloquium 2018: Bidang Teknik dan Rekayasa. The 7th University Research Colloquium 2018: Bidang Teknik dan Rekayasa*, Surakarta, Indonesia: Universitas Muhammadiyah Surakarta, pp. 96–105. Available at: https://repository.urecol.org/index.php/proceeding/article/view/22 (Accessed: 11 August 2025).

Hirsch, J. (2011) 'Aluminium in Innovative Light-Weight Car Design', *Materials Transactions*, 52(5), pp. 818–824. Available at: https://doi.org/10.2320/matertrans.L-MZ201132.

Kumar, A.H. and Deepanjali, V. (2016) 'Design & Analysis of Automobile Chassis', *International Journal of Engineering Science and Innovative Technology (IJESIT)*, 5(1), pp. 187–196.

Lee, D.-C. and Han, C.-S. (2009) 'CAE (Computer Aided Engineering) Driven Durability Model Verification For The Automotive Structure Development', *Finite Elements in Analysis and Design*, 45(5), pp. 324–332. Available at: https://doi.org/10.1016/j.finel.2008.10.004.

Mat, M.H. and Ghani, A.R.Ab. (2012) 'Design and Analysis of "Eco" Car Chassis', *Procedia Engineering*, 41, pp. 1756–1760. Available at: https://doi.org/10.1016/j.proeng.2012.07.379.

Rao, K.S. *et al.* (2017) 'Design And Analysis Of Light Weighted Chassis', *International Journal of Mechanical Engineering and Technology (IJMET)*, 8(5), pp. 96–103.

Tisza, M. and Czinege, I. (2018) 'Comparative Study Of The Application Of Steels And Aluminium In Lightweight Production Of Automotive Parts', *International Journal of Lightweight Materials and Manufacture*, 1(4), pp. 229–238. Available at: https://doi.org/10.1016/j.ijlmm.2018.09.001.

Tsirogiannis, E.C., Stavroulakis, G.E. and Makridis, S.S. (2019) 'Electric Car Chassis for Shell Eco Marathon Competition: Design, Modelling and Finite Element Analysis', *World Electric Vehicle Journal*, 10(1), p. 8. Available at: https://doi.org/10.3390/wevj10010008.

Wibowo, M.Y. *et al.* (2022) 'Perancangan Chassis Prototype Mobil Warak dan Simulasi Statik dengan Metode Finite Element Analysis', *Jurnal Mekanik Terapan*, 3(3), pp. 86–92. Available at: https://doi.org/10.32722/jmt.v3i3.5138.

Zamzam, O. *et al*. (2025) 'Structural Performance Evaluation Of Electric Vehicle Chassis Under Static And Dynamic Loads', *Scientific Reports*, 15(1), p. 5168. Available at: https://doi.org/10.1038/s41598-025-86924-w.