Jurnal Integrasi
Vol. 17 No. 1, April 2025, 59-69
e-ISSN: 2548-9828

Article History
Received January 2025
Accepted April 2025

Performance Comparison of Wheeled Soccer Robot Frames Using Finite Element Analysis (FEA)

Anugerah Wibisana[1], Muhammad Imron Shodiq[1], Muhammad Naufal Airlangga Diputra[2] and Naurah Nazhifah[1]

[1,2]Politeknik Negeri Batam
[1]Robotics Engineering Technology Study Program
[2]Mechatronics Engineering Study Program
[1,2]Ahmad Yani Street, Batam Centre, Batam 29461, Indonesia
E-mail: wibisana@polibatam.ac.id

Abstrak

Analisis rangka sangat penting dalam Kontes Robot Sepak Bola Indonesia Beroda (KRSBI-B) karena membantu mengidentifikasi kelemahan desain dan memungkinkan optimalisasi sebelum robot fisik dibangun. Studi ini membandingkan tiga desain rangka: standar, standar yang dimodifikasi, dan rangka dari tim Tech United. Analisa menggunakan teknik Finite Element Analysis (FEA) di SolidWorks yang setiap desain diuji dengan beban vertikal (25 kg) dan beban horizontal (80 kg), mensimulasikan kondisi nyata dalam kompetisi. Rangka Tech United menunjukkan performa terbaik, dengan nilai tegangan 5,8 MPa (vertikal) dan 180,0 MPa (horizontal), perpindahan minimal sebesar 0,018 mm dan 1,806 mm, serta nilai Faktor Keamanan (FOS) sebesar 9,5 dan 0,3. Rangka ini juga menunjukkan kekakuan, stabilitas, dan zona kerusakan terkendali yang lebih baik, di mana deformasi terkonsentrasi di area tertentu untuk menyerap benturan. Karakteristik ini menjadikan rangka Tech United sebagai pilihan paling tahan lama dan andal untuk meningkatkan performa robot dan ketahanannya dalam kompetisi.

Keywords: Analisis Elemen Hingga, Faktor Keamanan, Kontes Robot Sepak Bola Indoneia Beroda

Abstract

Frame analysis plays a vital role in the Indonesian Wheeled Soccer Robot Contest (KRSBI-B) by identifying design weaknesses and enabling optimization before building the physical robot. This study compares three frame designs: standard, modified standard, and the Tech United team's frame. Using Finite Element Analysis (FEA) in SolidWorks, each design is tested under vertical (25 kg) and horizontal (80 kg) loads, simulating real competition conditions. The Tech United frame shows the best performance, with stress values of 5.8 MPa (vertical) and 180.0 MPa (horizontal), minimal displacements of 0.018 mm and 1.806 mm, and FOS values of 9.5 and 0.3, respectively. It also displays superior stiffness, stability, and a controlled damage zone where deformation is intentionally localized to absorb impact. These characteristics make the Tech United frame the most durable and reliable option, improving overall robot performance and ensuring it can withstand the demands of competition.

Keywords: Finite Element Analysis, Factor of Safety, Wheeled Soccer Robot

1. Introduction

The Indonesia Robot Contest is an annual robotics competition in Indonesia with several categories, one of which is the Indonesian Wheeled Soccer Robot Contest, based on the international competition RoboCup Middle Size League as the initiator. This competition allows students to develop student abilities in various fields, such as mechanics, manufacturing, electronics, and programming. The competition demands the wheeled robots like football players to coordinate, play, and score autonomously, like human football in general [1], [2].

When design the robot for the competition, the weight of the components and robot base often creates vertical load pressure that affects the condition of the robot frame, both in static and dynamic conditions, and the enemy's robot may collide with the robot that will creates horizontal load pressure which poses a risk of stress and material displacement. Because of this, the developed robot must have high resistance, especially in the foundation or frame section of the robot, which is an important factor in supporting the frame to keep the frame prevent damage and changes in shape due to impact under a vertical and horizontal load pressure [3]. Therefore, this research aims to design a robot frame that is as strong as possible and can withstand the vertical load pressure of components and the risk of impact caused by the horizontal load pressure as effectively as possible.

The robot frame design was designed using SolidWorks 2020, which supports precision design and allows structural strength analysis with the Finite Element Analysis (FEA) feature [4]. FEA is a numerical analysis method that divides the design into small elements, called nodes, through the meshing process. In this study, the static analysis feature in SolidWorks was used to evaluate the frame's structural strength [5], [6], [7].

This analysis enables virtual testing of various robot frame designs to identify and address weaknesses before implementation on physical robots. Through optimization, the frame is expected to resist excessive stress that could lead to deformation or structural failure during matches, while ensuring a sufficient safety level based on the Factor of Safety (FOS) from simulation results [7], [8], [9], [10].

The simulation uses aluminum 6061 as the material and compares three frame designs: the current standard frame, a modified version of the standard frame, and the Tech United team's design from Eindhoven Team [11]. The results of this analysis are expected to contribute to the optimization of the robot design, both in terms of durability and material efficiency.

2. Method

This research begins with the design of a wheeled soccer robot frame to implement an optimal and efficient frame design during the competition. Figure 1 presents the research process flowchart, which consists of several stages described in the following sections.

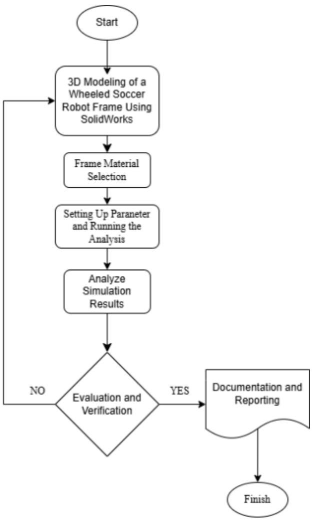

Figure 1: Flow diagram

A. 3D Modelling of Wheeled Soccer Robot Frame

The 3D modeling of the wheeled soccer robot frame begins with a design concept that meets the needs of the Indonesia Robot Contest, such as durability. The frame is designed to withstand movements that put excessive pressure on the structure. Therefore, the frame structure must have sufficient strength to support all robot components. Various frame designs are evaluated in this design process to find the best design. The 3D frame model design is made with the help of CAD software, namely SolidWorks 2020. The process of making this model involves determining the size and layout of each part of the structure, including joints. Each component is planned with strength, stability, and load distribution in mind to withstand intense use during the match.

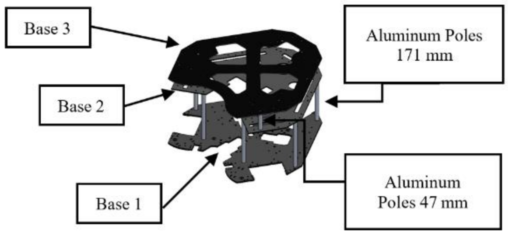

Figure 2: Isometric view of the Standard frame

The standard frame design shown in Figure 2 consists of three layers of aluminum base plates supported by nine aluminum poles. Six aluminum poles connect the first and second layers with a length of 171 mm, and the second and third layers are connected by 47 mm aluminum poles.

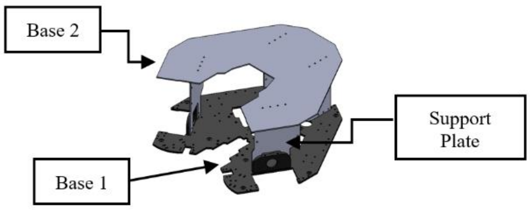

Figure 3: Isometric view of the modified frame

Figure 3 shows the modified frame design from a standard frame. The design uses supports from 4 aluminum plates connected to the second layer base plate using L-shaped aluminum angles.

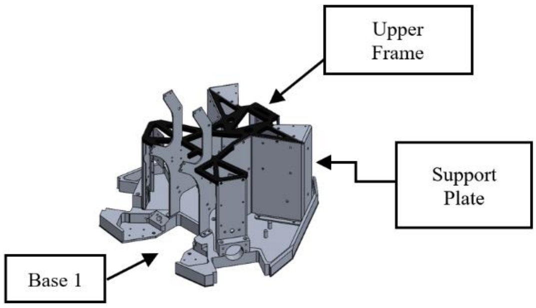

Figure 4: Isometric view of Tech United frame

Figure 4 shows Tech United's frame, which uses five aluminum plates as the top frame support.

B. Frame Material Selection

Choosing the right material is essential to balancing strength and weight in a robot frame. This research uses aluminum 6061, known for its high strength, durability, and lightweight properties. The excellent strength-to-weight ratio and machinability make the material ideal for structural applications like robot frames. Compared to aluminum 5052, aluminum 6061 has higher tensile and yield strength, allowing the material to handle greater loads and stress without deforming. While 5052 is more corrosion-resistant [12], 6061’s superior strength makes material better suited for high-performance robot designs.

SolidWorks data for 6061 aluminum shows key material properties. With a modulus of elasticity of 69,000 N/mm², the material is stiff and does not deform easily under pressure. A Poisson's ratio of 0.33 indicates moderate lateral contraction under axial compression. A shear modulus of 26,000 N/mm² indicates the material's strength under shear loads. The density of 6061 aluminum is 2,700 kg/m³, making the material a lightweight yet strong material. 6061 aluminum has a high tensile strength of 124,084 N/mm² and a yield strength of 55.1485 N/mm², making this capable of withstanding heavy loads without breaking. A coefficient of thermal expansion of 2.4e-05/K indicates that it will expand moderately with temperature changes and 170 W/(m K) thermal conductivity means the material can dissipate heat efficiently. With a specific heat of 1300 J/(kg·K), the material requires much energy for temperature changes, which is important for thermal stability. Additional information shows that aluminum 6061 is very suitable for various mechanical and structural uses; the material properties can be seen in Table 1[13].

TABLE I

ALUMINUM 6061 MATERIAL PROPERTIES

Properties	Value	Units
Elastic Modulus	69000	N/mm^2
Poisson's Ratio	0.33	N/A
Shear Modulus	26000	N/mm^2
Mass Density	2700	Kg/m^3
Tensile Strength	124,084	N/mm^2
Yield Strength	55.1485	N/mm^2
Thermal Expansion Coefficient	2.4e-05	/K

C. Setting Up Parameters and Running the Analysis.

To assess the performance of each frame design, simulations are conducted using SolidWorks Finite Element Analysis (FEA), focusing on stress, displacement, and factor of safety (FOS). FEA is a numerical method, solves engineering problems like stress analysis, heat transfer, and fluid flow by dividing structures into smaller elements for precise evaluation [14].

In each simulation, the following parameters are applied: the first parameter involves inputting the material properties into the simulation, which is set as alloy 6061. The second parameter involves applying external loads, which in this simulation are divided into two types, vertical load pressure and horizontal load pressure. The vertical load pressure input is set at 5 kg, derived from the original weight of the upper base, which will be maximized by multiplying by five so that the weight input in the simulation is 25 kg.

The second simulation results on each frame are carried out of the maximum weight of the wheeled robot soccer on the national competition rules, which is 40kg [2]. This will be multiplied by two, so the weight that the input is in the simulation is 80kg in the horizontal direction. This approach is used to evaluate the optimal strength of the frame when facing a vertical load pressure that is five times greater than normal conditions, and the horizontal load pressure two times greater than the maximum weight of the robot. The simulation results are visualized through SolidWorks through 3D models, graphs, diagrams, and data tables. The appearance of the upper base and the mass properties can be seen on Figure 5.

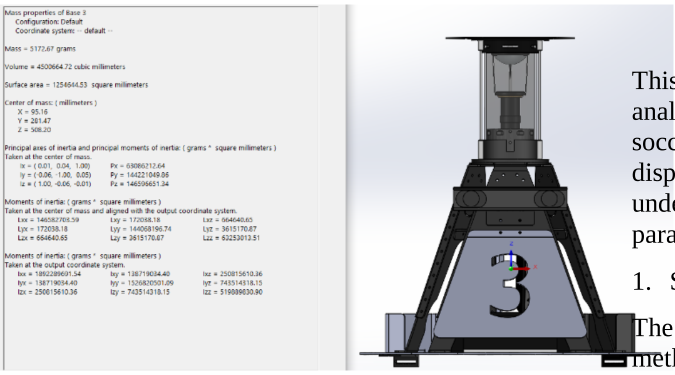

Figure 5: Left view of mass properties and the right view of the robot's top Base

The first simulation performed is stress analysis, which identifies the stress distribution on the frame under operational loads. This helps pinpoint high-stress areas that could lead to material failure. The stress occurring in an object can be formulated in Eq.1.

$$\sigma = \frac{F}{A} \qquad (1)$$

Where σ represents Stress or force per unit area (N/m²), F represents Force (N), and A represents Cross-sectional area (m²). The second simulation is displacement analysis, which evaluates how the frame deforms under load. Excessive displacement can compromise the robot's balance and stability during operation. This analysis ensures the frame maintains rigidity to maintain stability. The displacement occurring in an object can be formulated in Eq.2.

$$E = \frac{\sigma}{\varepsilon} \qquad (2)$$

Where E represents Modulus of elasticity, σ represents Stress (N/m²), and ε represents Strain. The third simulation is the factor of safety (FOS) analysis, used to assess the safety margin of the frame under load. FOS represents the ratio between the applied stress and the material's strength before permanent deformation. This analysis highlights areas with low safety margins, guiding designers to modify or strengthen components to ensure reliability[15][16]. The FOS occurring in an object can be formulated in Eq.3.

$$\eta = \frac{sy}{\sigma e} \qquad (3)$$

Where Ƞ represents Factor of Safety, sy represents Yield strength of the material (N/m²), and σe represents Maximum von Mises stress.

3. Results and Discussion

This chapter presents the simulation results and analysis of three different frame designs for a wheeled soccer robot. The evaluations include stress, displacement, and Factor of Safety (FOS) analyses under vertical and horizontal load pressures with set parameters.

1. Stress analysis

The first test simulated is stress analysis, which is a method to determine the frame's response to static loading by measuring the stress that occurs in the structure. The results of the analysis per frame from the simulation shown in Figure 6.

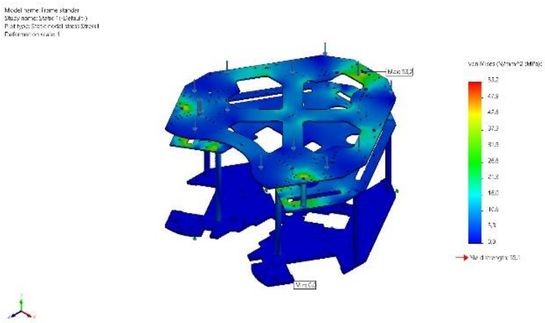

Figure 6: Vertical load stress analysis results of the standard frame

Figure 6 shows the frame highest stress level, 53.2 MPa, caused by the vertical load pressure. This shows that the standard frame is limited in distributing the load evenly, resulting in higher stress concentrations. The significant stress in this design indicates the possibility of distortion or danger when used in heavier load situations.

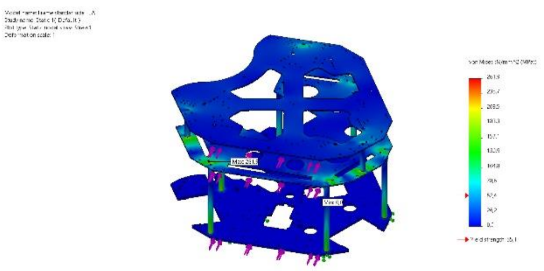

Figure 7: Horizontal load stress analysis results of the standard frame

In Figure 7, the frame shows the stress level at 261.9 MPa, caused by the horizontal load pressure, The results demonstrate that the standard frame faces challenges in handling horizontal loads effectively, as the frame exhibits high stress concentrations. This significant stress suggests a heightened risk of deformation or failure when exposed to heavier horizontal loads or extreme operating conditions.

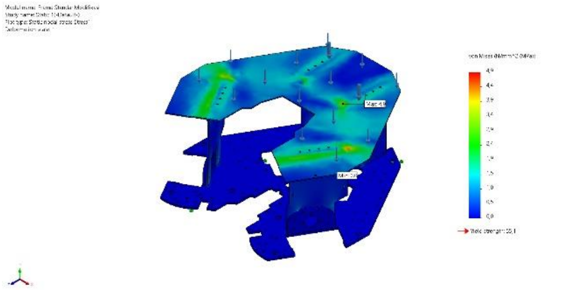

Figure 8: Vertical load stress analysis results of the modified standard frame

Figure 8 shows the results of the modified standard frame. Tests show a stress of 4.9 MPa from the vertical load pressure. This frame reduces the stress level, resulting in better weight distribution, increased resistance to deformation, and frame stability in challenging operational environments.

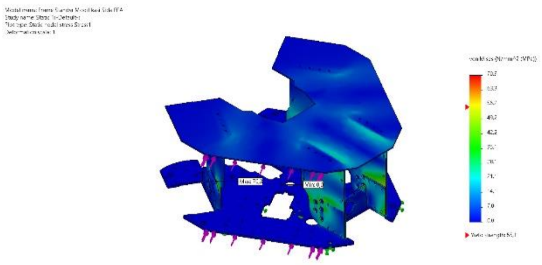

Figure 9: Horizontal load stress analysis results of the modified standard frame

Figure 9 shows a stress of 70.3 MPa from the horizontal load pressure, indicating improved weight distribution. This design enhances resistance to deformation and ensures greater frame stability, making the frame more reliable in demanding operational environments.

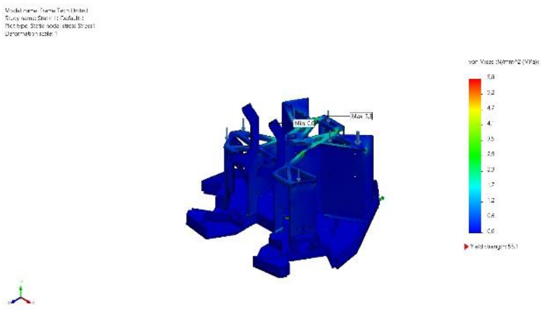

Figure 10: Vertical load Results of stress analysis Tech United frames

In Figure 10, the Tech United frame shows a stress of 5.8 MPa caused by the vertical load pressure, which shows satisfactory load-bearing capacity performance.

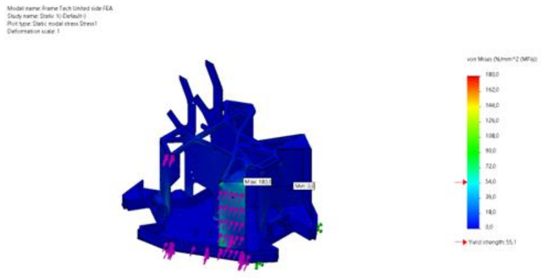

Figure 11: Horizontal Load Results of stress analysis of Tech United frames

Figure 11 shows a stress of 180.0 MPa caused by the horizontal load pressure. Although not as effective as the modified standard frame in distributing the load, the stress result caused by the horizontal load pressure in the frame tech united shows that the Tech United design offers increased durability and can maintain the stability of a solid structure and a focused distributed load.

TABLE II

VERTICAL STRESS ANALYSIS RESULT DATA

Frame	Value	Units
Standard Frame	53.2	N/mm^2 (MPA)
Modified Standard Frame	4.9	N/mm^2 (MPA)
Frame Tech United	5.8	N/mm^2 (MPA)

As shown in Table II, the summarized data from the vertical stress analysis indicate that differences in frame design significantly impact the load-bearing capacity. The modified frame and the Tech United frame demonstrate better stress distribution compared to the standard frame.

TABLE III

HORIZONTAL STRESS ANALYSIS RESULT DATA

Frame	Value	Units
Standard Frame	261.9	N/mm^2 (MPA)
Modified Standard Frame	70.3	N/mm^2 (MPA)
Frame Tech United	180.0	N/mm^2 (MPA)

Table III summarizes the data from the horizontal stress analysis, revealing that the modified standard frame exhibits the lowest stress among the designs due to its focus on improved load distribution. Meanwhile, the Tech United frame demonstrates a well-distributed load pressure, effectively controlling potential damage and maintaining structural integrity.

2. Displacement analysis

In this analysis, displacement measurements are conducted for each frame design to evaluate the rigidity and stability of the robot's frame under load. The displacement value indicates the extent to which the structure shifts or deforms when subjected to external forces. Both vertical and horizontal displacement values are analyzed to assess the frame's performance under different loading conditions. These results are critical to help identify designs that can effectively maintain the shape and integrity under vertical pressure from components and horizontal impacts from collisions, ensuring optimal performance and durability.

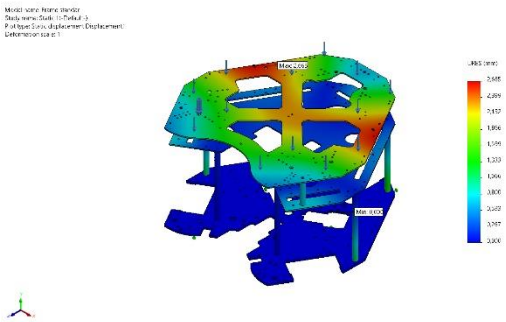

Figure 12: Vertical load displacement analysis results standard frame

In Figure 12, the current standard frame is loaded with vertical loads, and there is a reasonably visible change in position, with a displacement value of 2,655 mm on the standard frame. This displacement value shows that the standard frame has a minor stiffness, causing the frame to deform significantly when subjected to pressure. This shows the importance of refining the design to maintain stability.

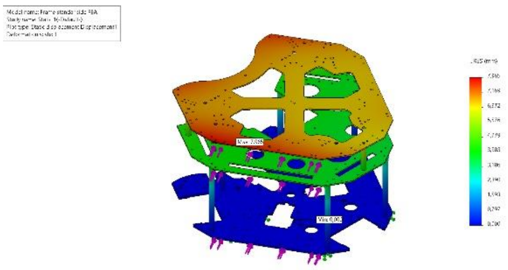

Figure 13: Horizontal Load displacement analysis results standard frame

Figure 13 shows the standard frame under horizontal loading, showing a noticeable positional shift with a displacement value of 7.965 mm. This significant deformation highlights the frame's lack of stiffness, making the frame prone to instability under pressure. These findings emphasize the need for design improvements to enhance rigidity and ensure structural stability.

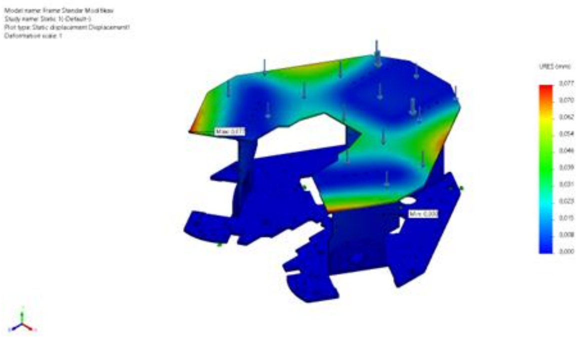

Figure 14: Vertical load displacement analysis results of modified standard frame

In Figure 14, the modified standard frame has a displacement of 0.077 mm. This shows a significant increase in stiffness compared to the standard frame. The decreased displacement values indicate that the modified design can reduce displacement and increase resistance to deformation while maintaining structural stability under load.

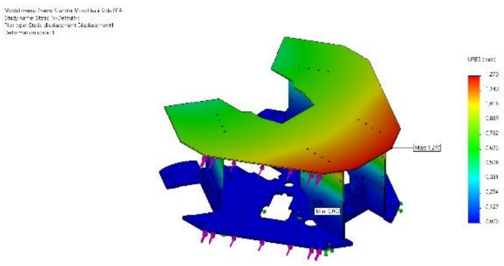

Figure 15: Horizontal load displacement analysis results of modified standard frame

Figure 15 shows that the modified standard frame exhibits a displacement of 1.270 mm, demonstrating a notable improvement in stiffness over the standard frame. The reduced displacement values suggest that the modified design effectively minimizes deformation, enhancing the frame's ability to resist displacement and maintain frame structural integrity under load. This indicates a stronger but with a note that the displacement point is so much distributed to the whole frame, which can cause permanent damage to the frame.

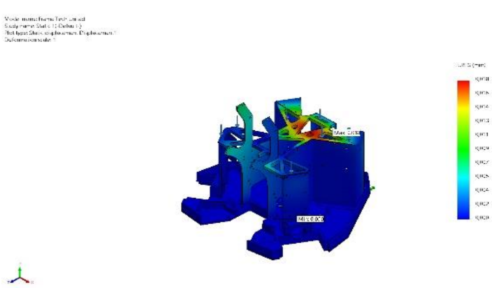

Figure 16: Vertical load displacement analysis results of Frame

Tech United

In Figure 16, the Tech United frame has the smallest displacement among the three designs; the result of the frame analysis is only 0.018 mm, that caused by the vertical load. This shows that the Tech United frame is the stiffest, with an extraordinary capacity to maintain the frame shape and stability under pressure. This small displacement makes the frame an excellent choice for designs that need to resist deformation.

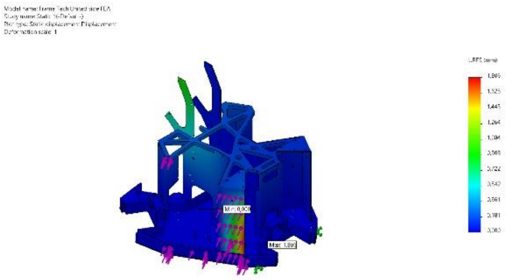

Figure 17: Horizontal load displacement analysis results of Frame Tech United

In Figure 17, the Tech United frame shows a displacement of 1.806 mm caused by horizontal load, placing the frame in the middle range compared to the other designs. However, despite this, the frame is designed to localize damage on the bent side plate, ensuring the primary structure remains unaffected. This focused distribution of displacement helps protect the core integrity of the frame while still allowing it to absorb impacts effectively, making the frame a reliable choice for maintaining stability under stress.

TABLE IV

VERTICAL LOAD DISPLACEMENT ANALYSIS RESULT DATA

Frame	Value	Units
Standard Frame	2.655	mm
Modified Standard Frame	0.077	mm
Frame Tech United	0.018	mm

In Table IV, the displacement results show that the Tech United frame outperforms the other designs in terms of stiffness under vertical load, demonstrating better stability. The displacement results show that the Tech United frame outperforms the other designs in terms of stiffness under vertical load, demonstrating better stability.

TABLE V

HORIZONTAL LOAD DISPLACEMENT ANALYSIS RESULT DATA

Frame	Value	Units
Standard Frame	7.965	mm
Modified Standard Frame	1.270	mm
Frame Tech United	1.806	mm

Additionally, for horizontal load, shown in Table V, the Tech United frame proves to be the safest, although the value is greater than the modified standard frame, because the frame effectively localizes damage without affecting the main structure.

3. Factor of safety (FOS) analysis

The Factor of Safety (FOS) is an important parameter in assessing a design's resistance to structural failure under a particular load. The value FOS indicates the frame's strength level in withstanding the load compared to the maximum capacity of the material before experiencing deformation or damage. The higher the value of FOS, the greater the margin of safety of the frame, so the frame will be more reliable and have better resistance to the risk of failure under various operational conditions.

For the standard frame, the analysis results from the vertical load pressure show that the minimum value of FOS is 1.0. This value is slightly above the minimum safe limit, indicating that the standard frame is on the verge of structural failure.

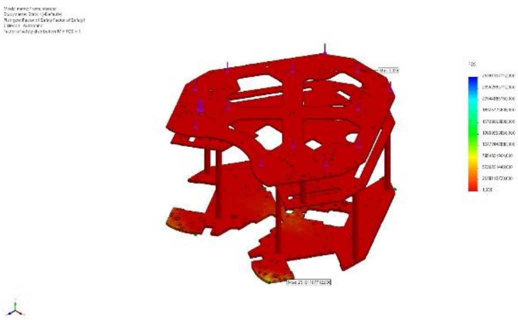

Figure 18: Vertical Load Results of standard frame analysis (FOS)

With a low FOS value, Figure 18 shows that standard frames require additional attention in the form of structural reinforcement or stronger alternative materials to increase frame durability. If improvements are not made, the risk of failure of these frames will remain high, especially in applications that require high performance and long-term durability. And using the calculation according to equation (3) can be obtained:

$$\text{Factor of safety} = \frac{55{,}15}{53.2} = 1.03$$

From this can be concluded that this simulation result is correct.

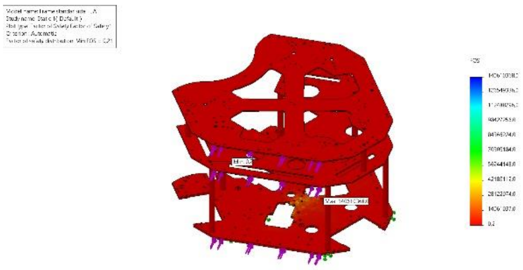

Figure 19: Horizontal Load Results of standard frame analysis (FOS)

Figure 19 shows that the Horizontal load pressure result is 0.2. This low FOS indicates that the standard frame has a fragile margin of safety, making the frame susceptible to damage. And within the calculation, according to equation (3), can be obtained:

$$\text{Factor of safety} = \frac{55.15}{261.9} = 0.21$$

So, if compared to the calculation can be confirmed that this simulation result is correct.

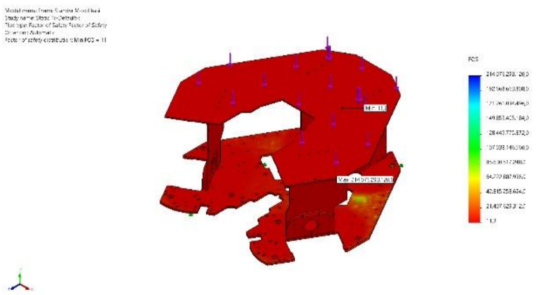

Figure 20: Results of analysis (FOS) of modified standard frame

Figure 20 shows that the modified standard frame shows that the minimum value of FOS from the vertical load increased significantly to 11.3. This high value FOS indicates that the modified frame has a much larger safety margin, providing excellent resistance to structural failure even under loads higher than normal operational loads. This makes the modified standard frame more reliable and durable than the standard frame. And using the calculation according to equation (3) can be obtained:

$$\text{Factor of safety} = \frac{55.15}{4.9} = 11.25$$

Comparing the simulation results with manual calculations confirms the accuracy.

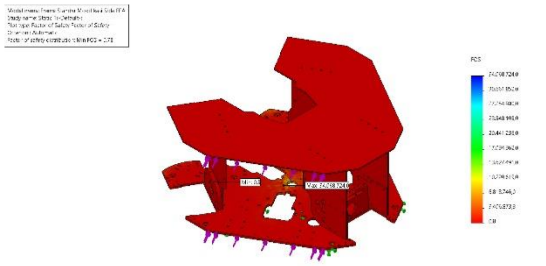

Figure 21: Results of analysis (FOS) of modified standard frame

Figure 21 shows the modified standard frame, where the minimum FOS for the horizontal load has significantly increased to 0.8. This improved FOS reflects a much greater safety margin, demonstrating the frame's exceptional ability to withstand structural failure even under elevated loads beyond normal operational conditions. As a result, the modified standard frame offers more durability compared to the standard frame. And from the calculation, according to equation (3), can be obtained:

$$\text{Factor of safety} = \frac{55.15}{70.3} = 0.78$$

Based on the calculation, the simulation output aligns well with the manual computations, validating the correctness.

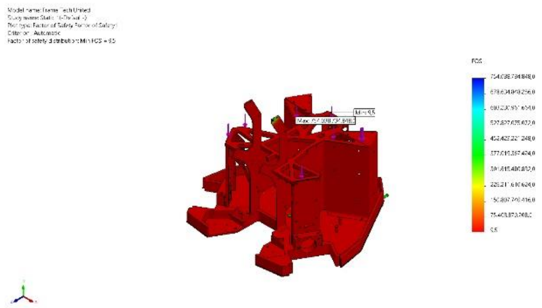

Figure 22: Vertical Load Results of analysis (FOS) of Tech United frame

Figure 22 shows that the Tech United frame design has a minimum value FOS of 9.5, showing the frame's ability to withstand loads with a fairly high safety margin. Although slightly below the modified standard frame, the Tech United frame still has excellent resistance to structural failure and can maintain structural performance under the vertical load conditions. And from the calculation, according to equation (3) can be obtained:

$$\text{Factor of safety} = \frac{55.15}{5.8} = 9.50$$

By matching the simulation data with manual calculations, this verified the accuracy of the results.

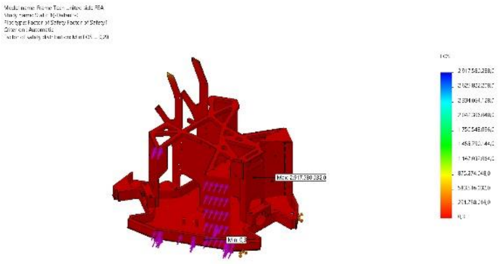

Figure 23: Horizontal Load Results of analysis (FOS) of Tech United frame

Figure 23 shows the Tech United frame design with a minimum FOS value of 0.3, indicating the frame's ability to endure horizontal loads with a modest safety margin. While slightly lower than the modified standard frame, the Tech United frame demonstrates strong resistance to structural failure and maintains reliable performance under horizontal load conditions. The lower FOS is attributed to the load concentration on the pre-bent sections, which effectively absorbs and redistributes the impact, enhancing the frame's overall load-handling efficiency. And within the calculation, according to equation (3), can be obtained:

$$\text{Factor of safety} = \frac{55.15}{180.0} = 0.30$$

The results from the simulation correspond to the manual calculations, proving the reliability.

TABLE VI

VERTICAL LOAD FOS ANALYSIS RESULT DATA

Frame	FOS
Standard Frame	1.0
Modified Standard Frame	11.3
Frame Tech United	9.5

Table VI shows the FOS analysis for the modified standard frame and Tech United, which have a higher safety margin than the standard frame, making them a safer choice for applications requiring resistance to heavy loads.

TABLE VII

HORIZONTAL LOAD FOS ANALYSIS RESULT DATA

Frame	FOS
Standard Frame	0.2
Modified Standard Frame	0.8
Frame Tech United	0.3

Table VII shows the FOS analysis for the modified standard frame and Tech United has a higher safety margin than the standard frame, making it a safer choice for applications requiring resistance to heavy loads.

TABLE VIII

VERTICAL LOAD ANALYSIS RESULT DATA

Frame Design	Stress	Displacement	FOS
Standard Frame	53.2 MPA	2,655 mm	1.0
Modified Standard Frame	4.9 MPA	0.077 mm	11.3
Frame Tech United	5.8 MPA	0.018 mm	9.5

Table VIII shows all the analysis results under vertical load pressure, with the Tech United frame outperforming the others. The frame achieves the lowest stress of 5.8 MPa, minimal displacement of 0.018 mm, and a high FOS of 9.5, demonstrating a focused load distribution, best stiffness, and a robust safety margin. This makes the Tech United frame the most reliable choice for durability and performance.

TABLE IX

HORIZONTAL ANALYSIS RESULT DATA

Frame Design	Stress	Displacement	FOS
Standard Frame	261.9	7.965 mm	0.2
Modified Standard Frame	70.3	1.270 mm	0.8
Frame Tech United	180.0	1.806 mm	0.3

Table IX shows all the analysis results under horizontal load pressure, where the Tech United frame stands out. The frame achieves a moderate stress of 180.0 MPa, controlled displacement of 1.806 mm, and an FOS of 0.3, with deformation concentrated in specific areas. These characteristics make the Tech United frame the most reliable option for maintaining stability and durability under horizontal loads.

4. Conclusions

This research evaluates the performance of three types of wheeled soccer robot frame designs: the standard frame, the modified standard frame, and the Tech United frame, using stress, displacement, and Factor of Safety (FOS) analyses under vertical 25 kg and horizontal 80 kg load pressures.

The results of the analysis show that the Tech United frame produced superior performance in terms of stress. Under the vertical load analysis, the frame showed a stress value of 5.8 MPa, while for the horizontal load analysis, the frame showed 180.0 MPa. These results highlight the frame's ability to handle significant loads, with reduced stress concentrations compared to other designs.

The displacement analysis reveals that the Tech United frame exhibits exceptional rigidity, with a minimal displacement of 0.018 mm under vertical load and 1.806 mm under horizontal load. These values indicate that the frame is highly resistant to deformation, ensuring better structural stability under both vertical and horizontal load conditions.

Regarding the Factor of Safety (FOS), the Tech United frame demonstrates a value of 9.5 under vertical load, indicating a significant safety margin. However, under horizontal load conditions, the FOS is 0.3, which is lower but still within acceptable limits, showcasing the frame's capacity to withstand forces while prioritizing controlled deformation in certain areas.

Additionally, the Tech United frame demonstrates the highest stiffness, exceptional stability, and a controlled damage zone where deformation is concentrated in specific areas designed to absorb the impact. This feature enhances the frame's durability and reliability, making this frame the best choice for optimizing the robot's performance and ensuring its robustness under competitive conditions.

Acknowledgment

The author would like to extend our heartfelt gratitude to the Barelang 63 team for their unwavering support and primary contributions to this research. Their dedication, expertise, and teamwork were essential in ensuring the success of this project. We also wish to thank the BRAIL members for their valuable support throughout the process. Their assistance has been instrumental in refining and enhancing the quality of this study. We are deeply appreciative of both teams' efforts in making this work possible.

References

[1] T. Balch, T. Schmitt, F. Schreiber, and B. Cunha, "Middle Size Robot League Rules and Regulations for 2009," 2009.

[2] B. Kusumoputro et al., "Pedoman Kontes Robot Indonesia (Kri) Pendidikan Tinggi Tahun 2024," Kementeri. Pendidikan, Kebudayaan, Riset, dan Teknol., pp. 1–164, 2024.

[3] I. G. Wiratmaja, N. A. Wigraha, and K. Purnayasa, "Analisis Tegangan Statik Dan Deformasi Frame Electric Ganesha Scooter Portable (E-Gaspol) Menggunakan Software Solidworks," Otopro, vol. 19, no. 1, pp. 8–17, 2023, doi: 10.26740/otopro.v19n1.p8-17.

[4] A. Welch-Phillips, D. Gibbons, D. P. Ahern, and J. S. Butler, "What Is Finite Element Analysis?," Clin. Spine Surg., vol. 33, no. 8, pp. 323–324, 2020, doi: 10.1097/BSD.0000000000001050.

[5] M. F. Arliansyah et al., "Analisa Finite Element Method (FEM) Uji Beban Pada Meja Polyethylene," J. Jalasena, vol. 4, no. 2, pp. 122–125, 2023.

[6] I. Dumyati and S. Nurhaji, "Modeling dan Simulasi Finite Element Analysis pada Segitiga T Sepeda Motor Menggunakan Sofware Ansys 2023," Quantum Tek. J. Tek. Mesin Terap., vol. 5, no. 1, pp. 26–30, 2023, doi: 10.18196/jqt.v5i1.19012.

[7] I. Muhlisin and S. Sudiman, "Pengaruh Variasi Beban terhadap Faktor Kekuatan Rangka Sepeda dari Bahan AISI 1035 Steel (SS) dengan Simulasi Solidworks," Briliant J. Ris. dan Konseptual, vol. 9, no. 1, p. 236, 2024, doi: 10.28926/briliant.v9i1.1822.

[8] M. Y. Wibowo, I. Maulana, A. A. Ghyferi, B. A. Kurniawan, and M. Nuril, "Perancangan Chassis Prototype Mobil Warak dan Simulasi Statik dengan Metode Finite Element Analysis," J. Mek. Terap., vol. 3, no. 3, pp. 86–92, 2022, doi: 10.32722/jmt.v3i3.5138.

[9] Sandy Suryady and Eko Aprianto Nugroho, "Simulasi Faktor Keamanadan Pembebanan Statik Rangka Pada Turbin Angin Savonius," J. Ilm. Multidisiplin, vol. 1, no. 2, pp. 42–48, 2022, doi: 10.56127/jukim.v1i2.94.

[10] F. A. Budiman, A. Septiyanto, Sudiyono, A. D. N. I. Musyono, and R. Setiadi, "Analisis Tegangan von Mises dan Safety Factor pada Chassis Kendaraan Listrik Febrian Arif Budiman dkk / Jurnal Rekayasa Mesin," Rekayasa Mesin, vol. 16, no. 1, pp. 100–108, 2021.

[11] F. Schoenmakers et al., "Tech United Eindhoven Team Description 2013 - Middle Size League," vol. 2011, 2013, [Online]. Available: http://www.techunited.nl/media/files/TDP2013.pdf

[12] J. Arif, Pungkas Prayitno, and Halan Al Hafidh, “Analisis static pada aluminium 5052 dengan variasi sudut menggunakan solidworks,” TEKNOSAINS J. Sains, Teknol. dan Inform., vol. 10, no. 1, pp. 38–50, 2023, doi: 10.37373/tekno.v10i1.269.

[13] Matweb, “MatWeb, Your Source for Materials Information,” MatWeb, pp. 1–2, 2015, [Online]. Available: http://www.matweb.com/search/datasheet.aspx?MatGUID=ff6d4e6d529e4b3d97c77d6538b29693

[14] M. I. Himawan, “Analisis Kekuatan Tarik Baja Karbon Rendah Dengan Metode Elemen Hingga Menggunakan Software (Solidwork),” J. Ekon. Vol. 18, Nomor 1 Maret201, vol. 2, no. 1, pp. 41–49, 2020.

[15] P. Kurowski, “Engineering Analysis with SolidWorks Simulation 2019,” Eng. Anal. with SolidWorks Simul. 2019, 2019, doi: 10.4271/9781630572372.

[16] I. N. Agus Adi, K. R. Dantes, and I. N. P. Nugraha, “Analisis Tegangan Statik Pada Rancangan Frame Mobil Listrik Ganesha Sakti (Gaski) Menggunakan Software Solidworks 2014,” J. Pendidik. Tek. Mesin Undiksha, vol. 6, no. 2, p. 113, 2018, doi: 10.23887/jjtm.v6i2.13046.