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

Muhammad Imron Shodiq[1] * and Anugerah Wibisana[1]

[1] Department of Electrical Engineering, Batam State Polytechnic, Batam, Indonesia

* Email: m.imron@students.polibatam.ac.id

Abstract— Frame analysis is critical in the Indonesian Wheeled Football Robot Contest (KRSBI-B) because with this analysis, we can find out the weaknesses of the design and can optimize the design before it is applied to the real physical robot. This study evaluates three types of robot frame designs: standard frames, modified standard frames, and frames made by the Tech United team. Each design is evaluated using the Finite Element Analysis (FEA) technique on Solidworks software to assess stress, displacement, and Factor of Safety (FOS) with a load of 25 kg or equivalent to 5 times the robot's upper baseload. The results of the analysis show that the frame Tech United produced superior performance, showing a stress value of 5.8 MPa, a smaller displacement of 0.018 mm, and a FOS value of 9.5. Based on these findings, the frame Tech United is recommended as the right choice in frame design because it is stronger and more stable which has maximum stiffness and minimum displacement and also the best choice to improve the robot durability.

Keywords: ***Finite Element Analysis* , *Factor Of Safety* , wheeled soccer robot , aluminum 6061.**

I. INTRODUCTION

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

In the Indonesia Robot Contest, the weight of the components and robot base often becomes pressure that affects the condition of the robot frame, both in static and dynamic conditions, which causes the danger 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 load of all robot components. A strong frame not only supports the load of parts and components but also helps prevent damage and changes in shape due to impact [3]. Therefore, researchers aim to design a robot frame that is as strong as possible and can withstand the load of components and the risk of impact as effectively as possible.

The robot frame design was created using Computer-Aided Design (CAD) software 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 Solidworks, the feature used is static analysis [5], [6], [7].

This analysis aims to enable virtual testing of robot frame design variations to identify and correct design weaknesses before implementation in real physical robots. With this optimization, the frame is expected to avoid excessive stress that can cause deformation and structural failure during the game and ensure that the frame has an adequate level of safety with the Factor of Safety (FOS) generated from the simulation [8], [9], [10], [11].

The material used in the simulation is aluminum 6061 and three frame designs to be compared: the first frame is the standard frame used in current robots, the second frame is a modification of the standard design; and the third frame is the frame design of the Tech United team from Eindhoven [12]. The results of this analysis are expected to contribute to the optimization of the robot design, both in terms of durability and material efficiency.

II. METHOD

This research begins with designing a wheeled soccer robot frame to implement an optimal and efficient frame design during the competition. Figure 1 shows the flow diagram of this research process, which has several stages which will be explained as follows:

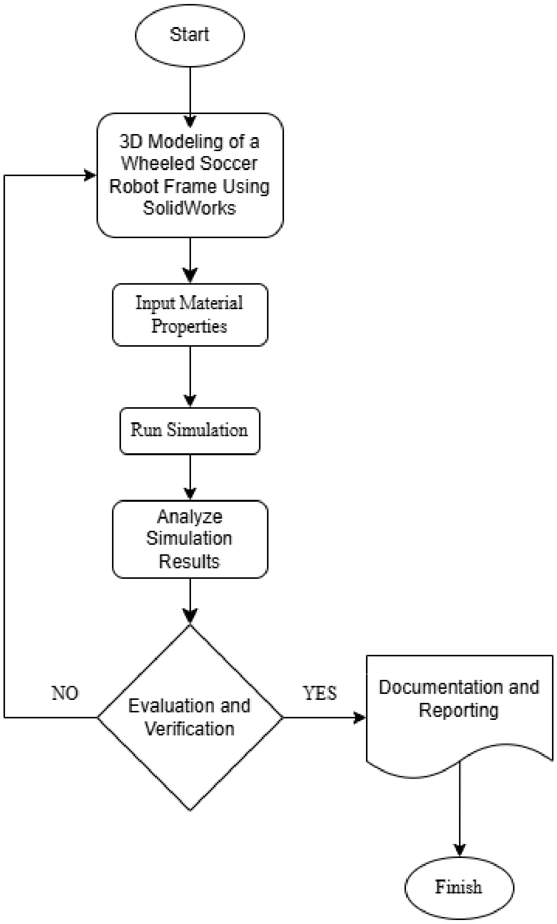

Fig. 1. Flow diagram

A. 3D Modeling of Wheeled Football 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.

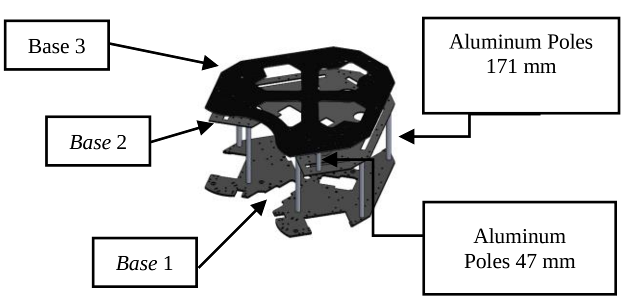

Fig. 2. Isometric view of Standard *frame*

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 for the second and third layers they are connected by 47 mm aluminum poles.

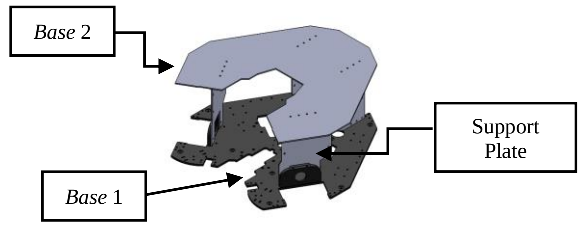

Fig. 3. Isometric view of Modified *frame*

Figure 3 show 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-shape angles aluminum.

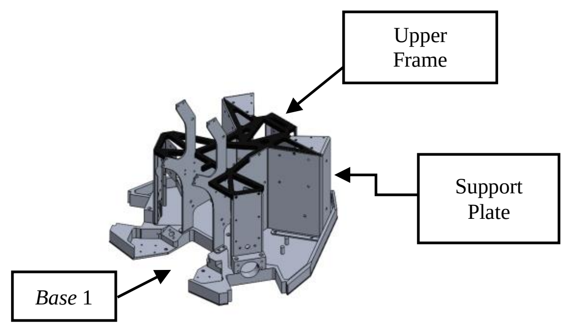

Fig. 4. Isometric view of Tech United *frame*

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

B. Frame Material Selection

Choosing the right material is crucial to achieving the perfect combination of strength and weight in a robot frame. This research conducts experimental simulation using aluminum 6061 which known for its strong tensile strength, durability, and lightweight properties. The mechanical properties of aluminum 6061, such as its high strength-to-weight ratio and excellent machinability make it suitable for structural applications that require strength and impact resistance such as robot frame design.

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 it 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 it capable of withstanding heavy loads without breaking. A coefficient of thermal expansion of 2.4e-05/K indicates that it expands moderately with temperature changes. Its 170 W/(m K) thermal conductivity means it can dissipate heat efficiently. With a specific heat of 1300 J/(kg·K), it 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. *Assessment Flow .*

To assess the performance of each frame design, simulations were performed using SolidWorks finite element analysis (FEA). Each design was subjected to stress, displacement, and factor of safety (FOS) analysis in three inspection categories.

The first category is stress analysis. This analysis is carried out to determine the stress distribution on the frame due to operational loads. This inspection is intended to identify high-stress areas that could potentially cause material failure. By reducing these stresses, the design can be strengthened in key areas to extend its service life.

The second category is displacement analysis. This analysis assesses how a structure moves or shifts under load. Too much distortion can cause the robot to lose balance and stability when moving. By studying these movements, designers can ensure the structure remains rigid for stability and prevent too much bending in the frame.

The third category is the factor of safety (FOS) analysis. The analysis is carried out to determine the frame safety factor and evaluate how well a design can withstand the load without failing. (FOS) is the relationship between the stress applied during the test and the strength of the material before undergoing permanent deformation. The (FOS) analysis provides details about areas with limited safety margins, allowing the designer to modify or strengthen certain components.

III. Results and Discussion

The simulation results on each frame are carried out by giving an external load of 5 kg from the original weight of the upper Base, which will be maximized by multiplying it by five so that the weight inputted in the simulation is 25 kg. This approach is used to evaluate the optimal strength of the frame when facing a load that is five times greater than normal conditions. The simulation results are visualized through SolidWorks through 3D models, graphs, diagrams, and data tables. The appearance of the upper Base can be seen on the right of Figure 5, and the mass properties can be seen on the left of Figure 5.

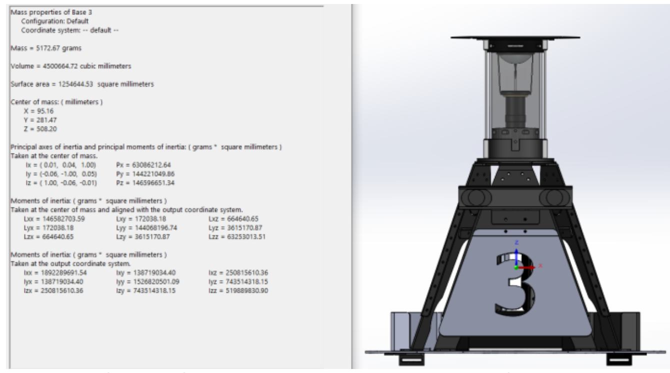

Fig. 5. Left view of *mass properties* and the right view of the robot's top *Base* .

A. *Stress analysis*

The first test simulated is stress analysis, which is a method to determine the frame response to static loading by measuring the stress (σ) that occurs in the structure. The results of the analysis per frame from the simulation are as follows:

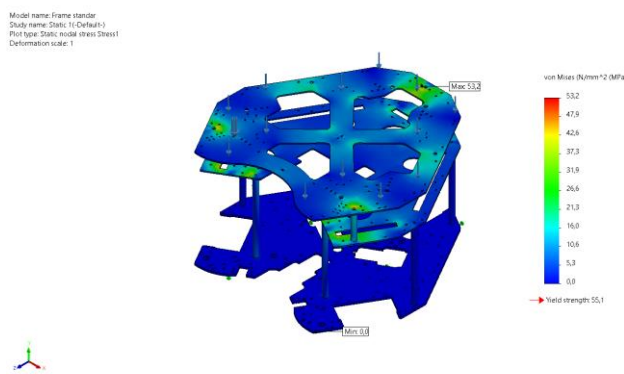

Fig .6. *Stress analysis results* standard *frame*

In Figure 6, this frame shows the highest stress level, 53.2 MPa. 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.

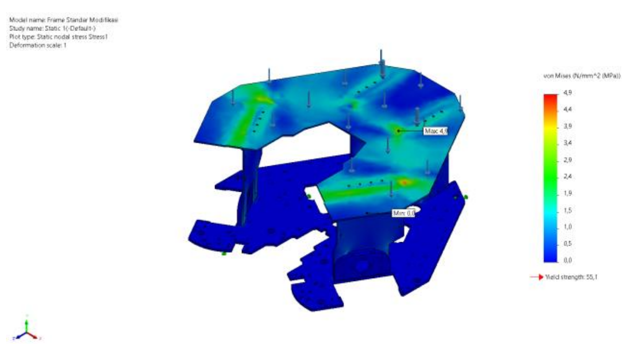

Fig.7. *Stress* analysis results standard modified *frame*

Figure 7 shows the results of the modified standard frame. Tests show a stress of 4.9 MPa on this frame. This frame reduces the stress level, resulting in better weight distribution, increased resistance to deformation, and frame stability in challenging operational environments.

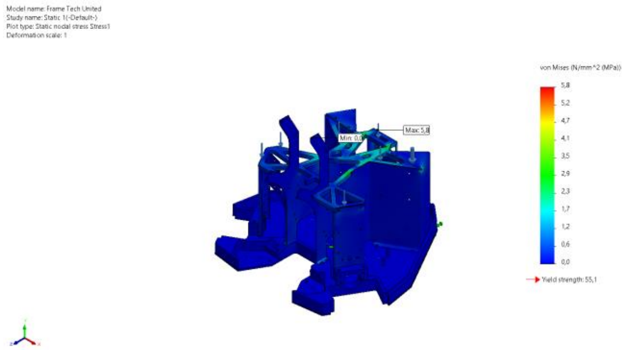

Fig. 8. Results of *stress analysis* Tech United *frames*

In Figure 8, the Tech United frame shows a stress of 5.8 MPa, which shows satisfactory load-bearing capacity performance. Although not as effective as the modified standard frame in distributing the load, the lower stress in the standard frame shows that the Tech United design offers increased durability and can maintain the stability of a solid structure.

TABLE II
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)

In Figure 9, the current standard frame is loaded with external 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 it to deform significantly when subjected to pressure. This shows the importance of refining the design to maintain stability.

B. Displacement analysis

In Figure 9, the current standard frame is loaded with external 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 it to deform significantly when subjected to pressure. This shows the importance of refining the design to maintain stability.

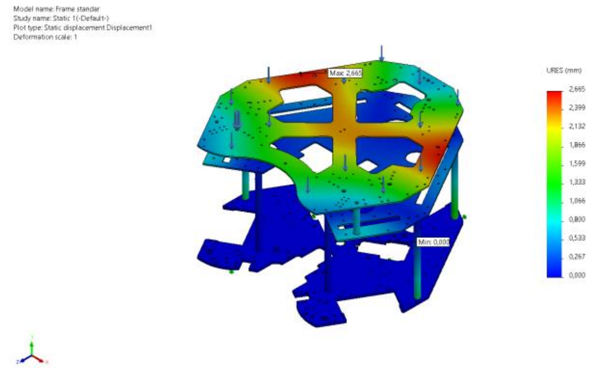

Fig. 9. *Displacement analysis results* standard *frame*

In Figure 9, the current standard frame is loaded with external 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 it to deform significantly when subjected to pressure. This shows the importance of refining the design to maintain stability.

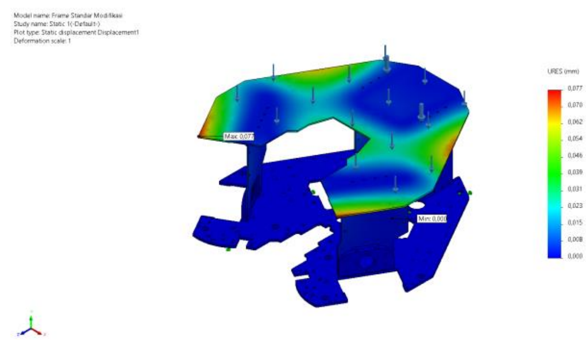

Fig. 10. *Displacement* analysis results standard modified *frame*

In Figure 10, 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.

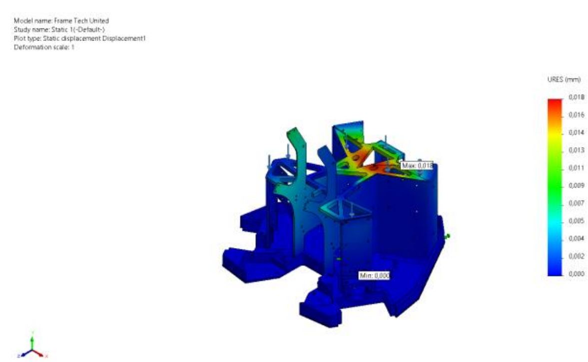

Fig. 11. *Displacement* analysis results *Frame* Tech United

In Figure 11, the Tech United frame has the most minor displacement among the three designs; the result of the frame analysis is only 0.018 mm. This shows that the Tech United frame is the stiffest, with an extraordinary capacity to maintain its shape and stability under pressure. This small displacement makes it an excellent choice for designs that need to resist deformation.

The displacement results show that the modified standard frame and the Tech United frame show superior stiffness compared to the standard frame, so they are more efficient in maintaining frame stability when subjected to loads. A recapitulation of the analysis values can be seen in Table 3.

TABLE III
DISPLACEMENT ANALYSIS RESULT DATA

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

C. 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 its material before experiencing deformation or damage. The higher the value 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.

The simulation results performed on each frame design provide an overview of the resistance of each design to heavy operational loads. For the standard frame, the analysis results show that the minimum value 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. This low FOS) indicates that the standard frame has a fragile margin of safety, making it susceptible to damage when used in situations involving heavy loads.

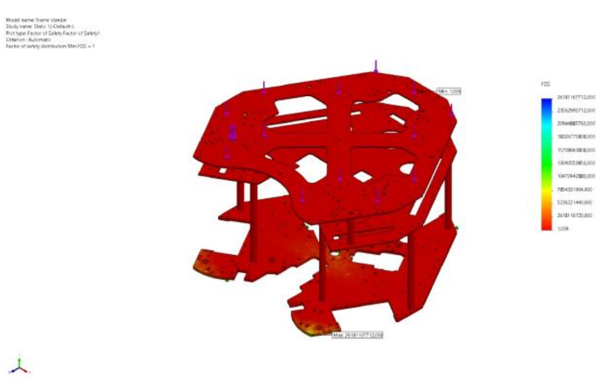

Fig. 12. Results of standard *frame analysis (FOS)*

With a low FOS value, Figure 12 shows that standard frames require additional attention in the form of structural reinforcement or stronger alternative materials to increase their 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.

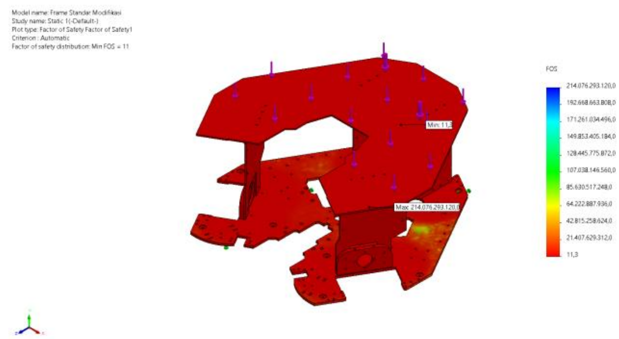

Fig. 13. Results of analysis (FOS) of modified standard *frame*

Figure 13 shows the standard modified frame shows that the minimum value (FOS) 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.

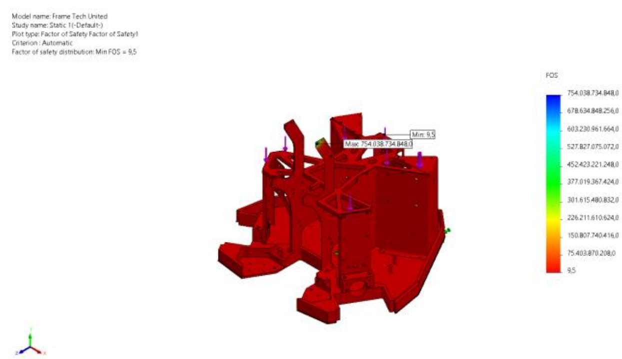

Fig. 14. Results of analysis (FOS) of Tech United *frame*

Figure 14 shows 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 is able to maintain structural performance under various load conditions. The FOS analysis shows that the modified standard frame and Tech United have a higher safety margin than the standard frame, making them a safer choice for applications requiring resistance to heavy loads.

TABLE 4
FOS ANALYSIS RESULT DATA

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

The following section summarizes the findings from the analysis of three soccer robot frame designs: a standard frame, a modified standard frame, and a frame created by the Tech United team. This study used SolidWorks software's Finite Element Analysis (FEA) technique to determine stresses, displacements, and Factor of Safety (FOS).

TABLE 5
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 5 shows comparison between the results of the stress, displacement, and FOS analysis on the three frame designs, thus providing a clearer understanding of how well each design can withstand operational loads.

The evaluation results show that each design exhibits different performance characteristics in load-bearing support. The standard frame exhibits higher stress levels and increased displacements than the modified design. The modified standard frame and the Tech United frame exhibit significant stress reductions, displacement reductions, and increases in (FOS) values. Both designs offer improved performance, demonstrating increased stiffness and structural stability.

IV. CONCLUSION

This research conducts experimental simulations to compare the performance of different frame designs for wheeled soccer robots. The experimental use frame designs of the standard frame of the current robot, the modified standard frame, and the Tech United frame. Three experimental experiments were carried out are stress, displacement, and Factor of Safety (FOS) analysis. The stress analysis results show the standard frame produces the highest stress at 53.2 MPA. In comparison, the modified standard frame produces 4.9 MPA and the Tech United frame produces 5.8 MPA, indicating better load distribution capabilities and reduced stress concentrations which can cause material damage. The displacement analysis shows the standard frame experienced the most significant displacement of 2,655 mm, indicating lower stiffness and a higher tendency for deformation when receiving loads. Meanwhile, the modified standard frame showed much smaller displacements of 0.077 mm, and the Tech United frame had the best stiffness of 0.018 mm, ensuring better structural stability. Regarding the Factor of Safety (FOS), the standard frame has the lowest (FOS) value of 1.0, which is close to the safe limit for material failure. In contrast, the modified standard frame and Tech United have higher (FOS) values of 11.3 and 9.5, indicating better safety margin and higher resistance to structural failure. Based on the stress, displacement, and FOS analysis results, the modified standard frame and Tech United are more recommended for use, with the Tech United frame being the best choice due to its highest stiffness and stability, making it an ideal choice for wheeled soccer robot design.

REFERENCE

[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.* , “Guidelines for the 2024 Indonesian Robot Contest (Kri) for Higher Education,” *Ministry of Education, Culture, Research, and Technology.* , pp. 1–164, 2024.

[3] IG Wiratmaja, NA Wigraha, and K. Purnayasa, “Static Stress Analysis and Deformation of Electric Ganesha Scooter Portable (E-Gaspol) *Frame Using* Solidworks Software,” *Otopro* , vol. 19, no. 1, pp. 8–17, 2023, doi: 10.26740/otopro.v19n1.p8-17.

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

[5] 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.

[6] MF Arliansyah *et al.* , “Finite Element Method (FEM) Analysis of Load Test on Polyethylene Table,” *J. Jalasena* , vol. 4, no. 2, pp. 122–125, 2023.

[7] I. Dumyati and S. Nurhaji, “Modeling and Simulation of *Finite Element Analysis* on Motorcycle T-Triangle Using Ansys 2023 Software,” *Quantum Tech. J. Tech. Applied Machine.* , vol. 5, no. 1, pp. 26–30, 2023, doi: 10.18196/jqt.v5i1.19012.

[8] I. Muhlisin and S. Sudiman, “The Effect of Load Variation on the Strength Factor of Bicycle *Frames* from AISI 1035 Steel (SS) Material with Solidworks Simulation,” *Briliant J. Ris. and Conceptual* , vol. 9, no. 1, p. 236, 2024, doi: 10.28926/briliant.v9i1.1822.

[9] MY Wibowo, I. Maulana, AA Ghyferi, BA Kurniawan, and M. Nuril, “Prototype Chassis Design of Warak Car and Static Simulation with *Finite Element Analysis Method* ,” *J. Mek. Terap.* , vol. 3, no. 3, pp. 86–92, 2022, doi: 10.32722/jmt.v3i3.5138.

[10] Sandy Suryady and Eko Aprianto Nugroho, “Simulation of Safety Factor and Static Loading *of Frame* on Savonius Wind Turbine,” *J. Ilm. Multidisiplin* , vol. 1, no. 2, pp. 42–48, 2022, doi: 10.56127/jukim.v1i2.94.

[11] FA Budiman, A. Septiyanto, Sudiyono, ADNI Musyono, and R. Setiadi, “Analysis of von Mises Stress and Safety Factor on Electric Vehicle Chassis Febrian Arif Budiman et al / Journal of Mechanical Engineering,” *Mechanical Engineering* , vol. 16, no. 1, pp. 100–108, 2021.

[12] 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

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