Analysis of the Structural Strength of the Barelang63 Wheeled Robot Goalkeeper Frame For Ball Impact and Robot Load Using Finite Element Analysis (FEA)

Muhammad Ribfli[1], Anugerah Wibisana[2], Tegar Adhi Nugraha Christ During[3]

[1,2,3]Politeknik Negeri Batam
muhammadribfli@gmail.com

Abstract. Wheeled soccer robots need a lightweight yet durable frame to ensure structural stability and consistent performance during competitions, where repeated collisions and ball impacts are inevitable. However, few studies have examined the structural behavior of such robot frames under realistic load conditions. This study analyzes the structural strength of the Barelang63 wheeled goalkeeper robot frame using Finite Element Analysis (FEA) in SolidWorks 2022. Three aluminum alloys—5052-H34, 6061, and 7075-T6—were tested under three loading scenarios: a horizontal collision force of 80 kgf, a vertical load of 28 kgf from robot components, and a ball impact force of 9.2 kgf. The simulation results indicated that Aluminum 6061 can safely withstand all loading conditions, with a Factor of Safety (FOS) above 1.0 and exceeding 2.0 for vertical and ball impact loads. Aluminum 7075-T6 showed the best overall performance, with the highest FOS and minimal displacement, while Aluminum 5052-H34 offered a good balance of strength, lightweight design, and cost efficiency. The findings provide practical guidance for optimizing lightweight robotic frame materials to improve durability and mechanical reliability in dynamic game environments.

Keywords: Soccer Robot, Barelang63, Finite Element Analysis, Factor Of Safety, Static

1 Introduction

The Indonesian Robot Competition (KRI) is an annual national event featuring several categories that promote innovation and technological progress in robotics. One of these categories is the Indonesian Wheeled Soccer Robot Competition (KRSBI Beroda), which is modeled after the international RoboCup Middle Size League. This event enables students to develop multidisciplinary skills in mechanics, manufacturing, electronics, and programming. Similar to human soccer, wheeled robots must operate autonomously, coordinate with each other, and score goals within specific rules and a maximum weight limit of 40 kg per robot [1], [2].

H. Widiastuti et al. (eds.), *Proceedings of the 8th International Conference on Applied Engineering (ICAE 2025)*, Advances in Engineering Research 290,
https://doi.org/10.2991/978-94-6463-982-7_31

During robot development, the combined weight of the base structure and internal components creates vertical loads acting on the robot frame under static conditions. Additionally, collisions with other robots during gameplay generate horizontal forces that can cause structural deformation or stress concentrations, potentially leading to material failure. Therefore, the robot frame must be designed to maintain high rigidity and strength to ensure durability and stability during operation [3].

Previous studies have examined the structural performance of various robot and vehicle frames using Finite Element Analysis (FEA), especially focusing on aluminum alloys such as 6061, known for their favorable balance of strength, machinability, and corrosion resistance. These studies show that optimizing geometry and material properties greatly impacts the Factor of Safety (FOS), displacement, and stress distribution in wheeled robotic frames [4].

This research aims to compare the mechanical performance of three aluminum alloys—5052-H34, 6061, and 7075-T6—for the Barelang63 wheeled goalkeeper robot frame using Finite Element Analysis (FEA). The goal is to identify materials that provide the best structural strength and durability while adhering to the 40 kg weight limit. The robot frame was modeled and analyzed with SolidWorks Simulation 2022, which offers precise stress and deformation evaluation through integrated meshing and solver tools [5][6][7].

Unlike most previous FEA-based studies on wheeled robot structures, this research specifically examines the Barelang63 goalkeeper robot, which faces unique loading conditions, including vertical loads from internal components, horizontal collision forces, and frontal ball impacts—factors rarely studied before. Incorporating these three realistic load cases offers a comprehensive view of actual competition scenarios. Additionally, manual analytical validation of FEA results improves the credibility of the numerical findings. These aspects highlight the originality and practical significance of this study in advancing the structural optimization of wheeled soccer robots [8][9][10][11].

2 Method

This study aims to determine the most effective and structurally robust material for the Barelang63 goalkeeper robot frame using Finite Element Analysis (FEA). The evaluation focuses on the frame's ability to withstand three primary loading conditions: horizontal collision forces, vertical loads from components, and frontal ball impacts. The overall research workflow, divided into several sequential stages, is illustrated in Figure 1.

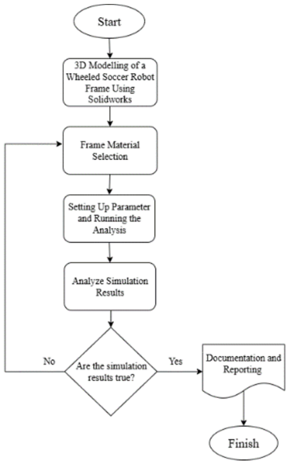

Fig. 1 Flow diagram

2.1 3D Modelling Frame Of Wheeled Soccer Robot

The initial stage involved designing the 3D model of the wheeled soccer robot frame according to the requirements of the Indonesian Robot Competition (KRI), focusing on mechanical strength, rigidity, and weight efficiency. The frame was modeled using SolidWorks 2022 CAD software, including key structural components like base plates, joints, and connection supports. Several preliminary configurations were tested to ensure optimal strength and load distribution while keeping the total frame weight within the 40 kg limit. The final Barelang63 goalkeeper robot frame model is shown in Figure 2.

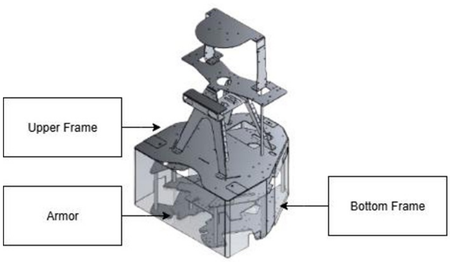

Fig. 2 Design Frame Goalkeeper

2.2 Frame Material Selection

When selecting the appropriate material for the robot frame, three aluminum alloys were evaluated: Aluminum 5052-H34, Aluminum 6061, and Aluminum 7075-T6. Aluminum 5052-H34 offers excellent corrosion resistance and good formability, with moderate strength (yield strength ≈ 215 MPa, tensile strength ≈ 260 MPa). Aluminum 6061 has lower strength (yield ≈ 55 MPa, tensile ≈ 124 MPa) but is highly machinable, weldable, and provides good thermal conductivity, making it suitable for frames requiring complex assembly. Aluminum 7075-T6 delivers the highest mechanical strength (yield ≈ 505 MPa, tensile ≈ 570 MPa), though it is heavier, more expensive, and less corrosion-resistant. The mechanical properties of these alloys—including density, tensile strength, yield strength, modulus of elasticity, and Poisson's ratio—are summarized in Table 1. These parameters are used as inputs in the Finite Element Analysis (FEA) to evaluate the stress distribution, deformation, and Factor of Safety (FOS) of the robot frame under simulated loading conditions.

Table 1 Material property

Property (Units)	Aluminium Alloy 5052-H34	Aluminium Alloy 6061	Aluminium Alloy 7075-T6
Elastic Modulus (N/mm^2)	70000	69000	72000
Poisson's Ratio (N/A)	0.33	0.33	0.33
Shear Modulus (N/mm^2)	25900	26000	26900
Mass Density (Kg/m^3)	2680	2700	2810
Tensile Strength (N/mm^2)	260	124.084	570
Yield Strength (N/mm^2)	215	55.1485	505
Thermal Expansion Coefficient (/K)	2.38e-05	2.4e-05	2.36e-05
Thermal Conductivity (W/(m·K))	137	170	130
Specific Heat (J/(kg·K))	880	1300	960

From a structural standpoint, the selection of material directly influences the robot's stiffness, energy absorption, and overall durability. Aluminum 5052-H34, while lightweight, exhibits higher ductility that may lead to greater deformation at joint regions during impact. Aluminum 6061 provides a balance between stiffness and manufacturability, making it an efficient option for lightweight frames that must endure repeated loading. Meanwhile, Aluminum 7075-T6, due to its high stiffness, minimizes deformation but may introduce localized stress concentrations at connection points. Thus, the material choice involves a trade-off between weight, strength, cost, and manufacturability, all of which significantly affect the robot's stability and mechanical reliability during competition.

2.3 Setting Up Parametersand Running the Analysis

Simulations were performed using SolidWorks Finite Element Analysis (FEA) to evaluate how different materials affect the same frame design, focusing on stress, displacement, and Factor of Safety (FOS). FEA divides complex structures into smaller elements to solve engineering problems such as stress, strain, and deformation numerically.

In each simulation, the initial step involves identifying the properties of Aluminum 5052-H34, 6061, and 7075-T6 materials within the SolidWorks FEA module. The subsequent step entails applying external loads to the frame, comprising vertical loads, horizontal loads, and ball impact loads. The vertical load, exerted downward on the upper frame, is set equal to the estimated total mass of the robot components, approximately 2.8 kgf as shown in Figure 3, multiplied by 10 to total 28 kgf. Laterally, a horizontal load of 80 kgf is applied to simulate a collision with another robot. Furthermore, a ball contact force of approximately 9.2 kgf is applied to the front frame; this force results from simulating a ball kick by a trained human [12]. The purpose of these load conditions is to emulate the stresses experienced during actual matches. Utilizing 3D models, contour maps, graphs, and tabular data, SolidWorks was used to present the simulation results, including stress distribution, deformation, and the Factor Of Safety (FOS).

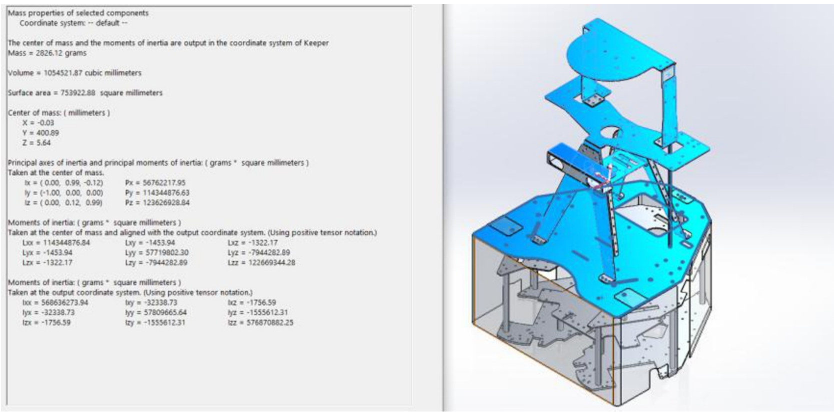

Fig. 3 Mass Distribution On Upper Base

Boundary conditions were applied as fixed geometry constraints at the wheel mounting points (Figure 4), fully restricting all degrees of freedom to simulate ground contact. The remaining frame structure was left free to deform, enabling the measurement of stress and displacement responses.

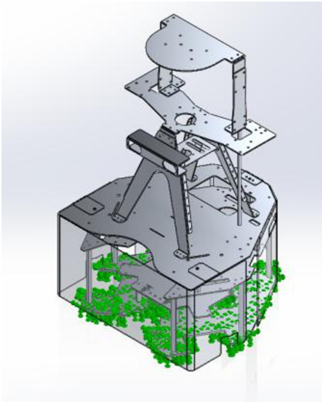

Fig. 4 Fixed Geometry for Boundary Condition

2.4 Meshing Process and Quality Assessment

The meshing process discretized the robot frame into finite elements to accurately capture load distribution. Solid tetrahedral elements were used because of their effectiveness in modeling complex 3 D geometries [13]. Mesh refinement was applied to critical regions such as joints, connection plates, and mounting points to identify potential stress concentration areas. The mesh quality was assessed using the 16- point Jacobian ratio check, which ranged from 1. 00 to 29.64. 64 and is within the recommended SolidWorks limit (≤40), indicating minimal distortion and good numerical stability.

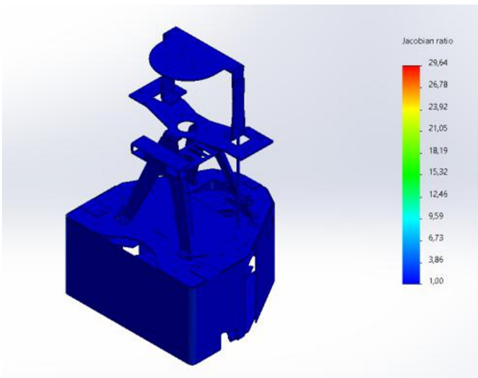

Fig. 5 Global Mesh Robot Frame

The generated mesh consisted of 168.826 solid elements and 353.966 nodes, with a minimum element size of 2.6 mm and a maximum element size of 52.0 mm. The mesh quality was categorized as High, and no distorted elements were detected, confirming mesh integrity. The percentage of elements with aspect ratio < 3 reached 85.5%, indicating that most elements possess good geometric proportions. These metrics verify that the mesh configuration provides a reliable balance between computational efficiency and accuracy in stress prediction.

The summary of mesh statistics, including total elements, nodes, aspect ratio, and quality indicators, is presented in Figure 6, which confirms that the mesh configuration satisfies SolidWorks Simulation's high-quality criteria and provides a stable basis for the subsequent FEA analysis.

Mesh Details

Study name	Static 1 (-Default-)
DetailsMesh type	Solid Mesh
Mesher Used	Blended curvature-based mesh
Jacobian points for High quality mesh	16 points
Max Element Size	52.0225 mm
Min Element Size	2.60112 mm
Mesh quality	High
Total nodes	335966
Total elements	168826
Maximum Aspect Ratio	50.777
Percentage of elements with Aspect Ratio < 3	85.5
Percentage of elements with Aspect Ratio > 10	0.457
Percentage of distorted elements	0
Number of distorted elements	0
Remesh failed parts independently	Off
Time to complete mesh(hh:mm:ss)	00:00:50
Computer name	

Fig. 6 Mesh Summary

To ensure numerical reliability, the mesh configuration was evaluated based on quality indicators rather than iterative refinement, due to computational constraints. The generated mesh exhibited a Jacobian ratio ranging from 1.00 to 29.64 (within the ≤40 SolidWorks criterion), 0% distorted elements, and over 85% of elements with an aspect ratio below 3, confirming good element uniformity. These parameters indicate a well-conditioned mesh suitable for static analysis.

According to comparable studies on aluminium structures analyzed using SolidWorks Simulation [14]. meshes of similar density and quality yield stress variations below 5%, which is considered acceptable for convergence. Therefore, the mesh employed in this study is deemed sufficiently converged for accurate stress and deformation evaluation while maintaining computational efficiency.

The first simulation run is a stress analysis, which determines the stress distribution on the frame under operational loads. This aids in identifying high-stress locations that may lead to material breakdown. The stress that occurs in an item can be stated in Equation 1.

$$S = \frac{F}{A} \tag{1}$$

This is where S stands for stress or force per unit area (N/m2), F for force (N), and A for cross-sectional area (m2). In the second simulation, displacement analysis, the frame's deformation under load is assessed. Over-displacement can jeopardize the stability and balance of the robot while it is in use. This analysis guarantees the stability of the frame by maintaining its rigidity. The displacement occurring in an object can be formulated in Equation 2.

$$\mathrm{M} = \frac{S}{\varepsilon} \tag{2}$$

In this case, M stands for modulus of elasticity, S for stress (N/m2), and ε for strain. To determine the frame's safety margin under load, the third simulation is the factor of safety (FOS) analysis. The ratio of the applied stress to the material's strength prior to irreversible deformation is known as FOS. In order to assure reliability, this analysis identifies regions with inadequate safety margins and directs designers to strengthen or change components [15][16]. Equation 3 allows for the formulation of the FOS that occurs in an object.

$$X = \frac{sy}{\sigma e} \tag{3}$$

Where X represents Factor of Safety, sy represents Yield strength of the material ($\mathrm{N/m^2}$),and σe represents Maximum von Mises stress

3 Result

In this chapter, three material comparisons for a wheeled soccer goalkeeper robot are analyzed and the simulation results are shown. Stress analysis, displacement, and Factor of safety under vertical load, horizontal load, and ball kick pressure to the robot frame with preset settings are all included in the evaluation.

3.1 Stress Analysis Horizontal Load Force

The first simulation test is the stress analysis, which evaluates the von Mises stress under horizontal loading. Figures 7–12 show the results for both the front and back armor components. The maximum stresses obtained were 0.1960 MPa in the front armor and 0.4629 MPa in the back armor for all three materials. The identical values occur because the applied load and frame geometry are the same, so the stress distribution does not depend on the material properties. When compared to the yield strengths of Aluminum 5052-H34 (≈ 215 MPa), Aluminum 6061 (≈ 55 MPa,), and

Aluminum 7075-T6 (≈ 505 MPa), all materials remain well below their limits. However, the margin of safety differs, with 7075-T6 providing the greatest safety, 5052-H34 performing adequately, and 6061 offering the least margin. This indicates that while the large frame geometry effectively minimizes stress, the choice of material still significantly influences the factor of safety.

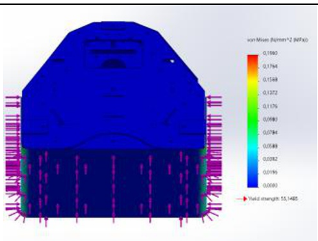

Fig. 7 Horizontal load stress analysis front armor results of 6061

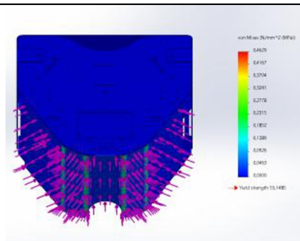

Fig. 8 Horizontal load stress analysis back armor results of 6061

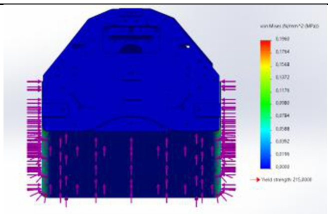

Fig. 9 Horizontal load stress analysis front armor results of 5052-H34

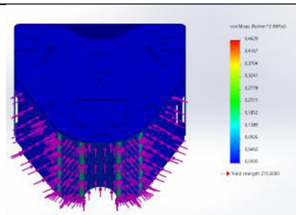

Fig. 10 Horizontal load stress analysis back armor results of 5052-H34

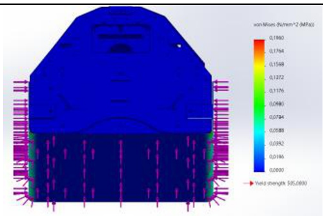

Fig. 11 Horizontal load stress analysis front armor results of 7075-T6

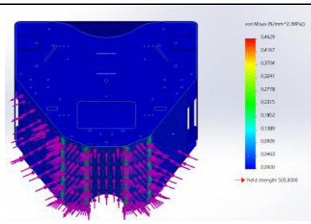

Fig. 12 Horizontal load stress analysis back armor results of 7075-T6

3.2 Displacement Analysis Horizontal Load Force

Following the stress analysis in Section 3.1, the displacement response of the frame under horizontal loading was evaluated, as shown in Figures 13–18. The results show that the maximum displacements in both the front and back armor were extremely small for all tested materials. In the front armor, the maximum displacements were 7.45e-06mm for Aluminium 5052-H34, 7.56e-06mm for Aluminium 6061, and 7.24e-06mm for Aluminium 7075-T6. In the back armor, the corresponding values were 1.603e-05mm, 1.626e-05mm, and 1.558e-05mm, respectively. When compared to the overall frame dimensions on the scale of several hundred millimeters these displacements are negligible. Such minimal deformation occurs because the frame's large geometry provides substantial stiffness and the applied loads are distributed across wide structural areas, thereby reducing localized bending. Among the three materials, Aluminium 7075-T6 produced the smallest displacement due to its higher elastic modulus, while Aluminium 6061 recorded the largest. However, the differences remain insignificant, confirming that the frame design maintains adequate rigidity regardless of material selection.

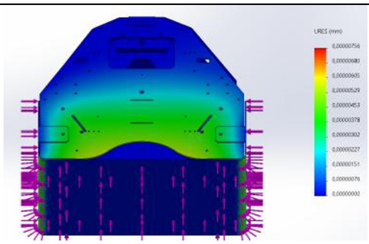

Fig. 13 Horizontal load displacement analysis front armor results of 6061

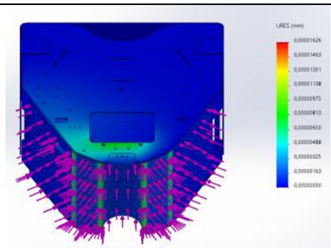

Fig. 14 Horizontal load displacement analysis back armor results of 6061

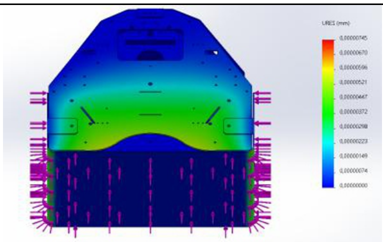

Fig. 15 Horizontal load displacement analysis front armor results of 5052-H34

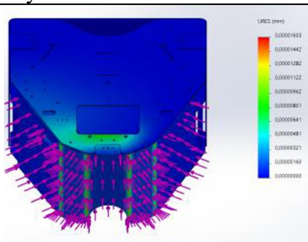

Fig. 16 Horizontal load displacement analysis back armor results of 5052-H34

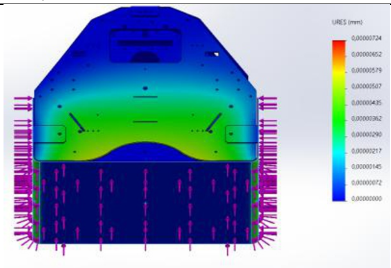

Fig. 17 Horizontal load displacement analysis front armor results of 7075-T6

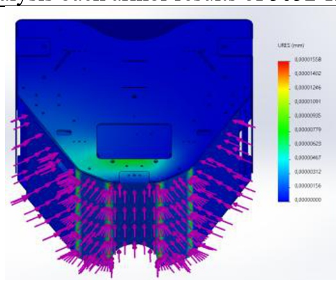

Fig. 18 Horizontal load displacement analysis back armor results of 7075-T6

3.3 Factor Of Safety Analysis Horizontal Load Force

The next stage of the analysis was the evaluation of the Factor of Safety (FOS), which represents the ratio between the yield strength of a material and the maximum stress it experiences. Figures 19–24 show the FOS distribution for the front and back armor components under horizontal loading. In the front armor, the FOS values obtained were 1096.87 for Aluminum 5052-H34, 281.35 for Aluminum 6061, and 2576.37 for Aluminum 7075-T6. In the back armor, the corresponding FOS values were 464.42, 119.12, and 1090.84. Since the maximum stresses were identical across all materials (0.1960 MPa in the front armor and 0.4629 MPa in the back armor), the differences in FOS are entirely due to variations in yield strength. Aluminum 7075-T6 exhibited the highest FOS, reflecting its superior strength, while Aluminum 6061 produced the lowest FOS due to its much lower yield strength in the SolidWorks database (≈ 55 MPa). Although all FOS values remained above 1.0, confirming that the frame is structurally safe, the margin of safety differs significantly, with 7075-T6 being the most reliable and 6061 offering the least margin.

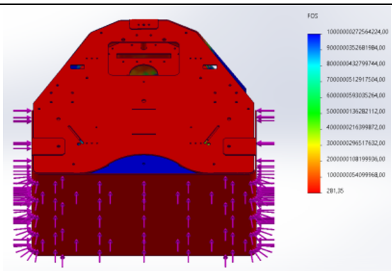

Fig. 19 Horizontal load FOS analysis front armor results of 6061

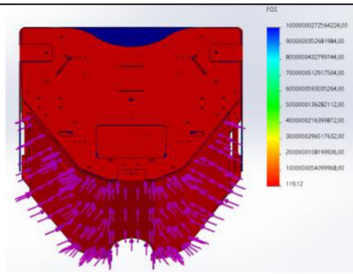

Fig. 20 Horizontal load FOS analysis back armor results of 6061

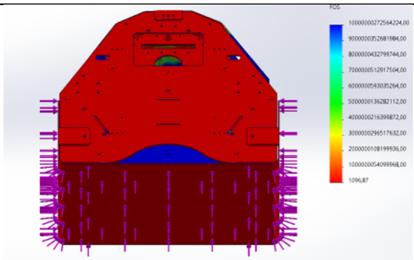

Fig. 21 Horizontal load FOS analysis front armor results of 5052-H34

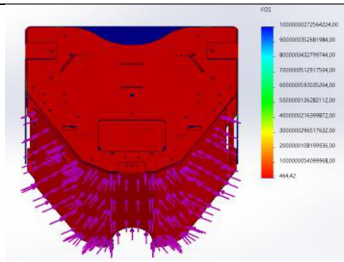

Fig. 22 Horizontal load FOS analysis back armor results of 5052-H34

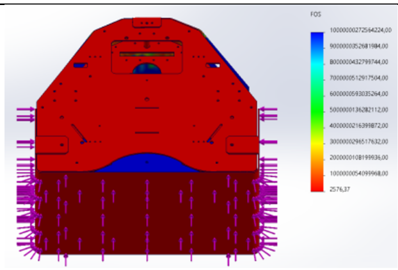

Fig. 23 Horizontal load FOS analysis front armor results of 7075-T6

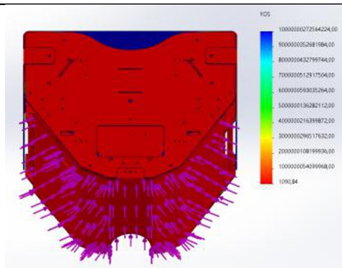

Fig. 24 Horizontal load FOS analysis back armor results of 7075-T6

3.4 Stress Analysis Vertical Load Force

The stress analysis under vertical loading shows that the maximum values were almost identical across the three materials, with Aluminum 7075-T6 reaching 39.08 MPa and both Aluminum 5052-H34 and Aluminum 6061 recording 39.09 MPa. When compared to their yield strengths, Aluminum 6061 (≈ 55 MPa) is close to its limit and at risk of yielding, while Aluminum 5052-H34 (≈ 215 MPa) and Aluminum 7075-T6 (≈ 505 MPa) remain safely below their thresholds. These results, in Figures 25–27, confirm that although the applied stresses are distributed evenly by the frame geometry, the material properties ultimately determine whether the structure can safely withstand vertical loads.

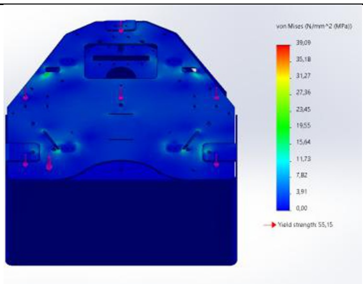

Fig. 25 Vertical load stress analysis results of 6061

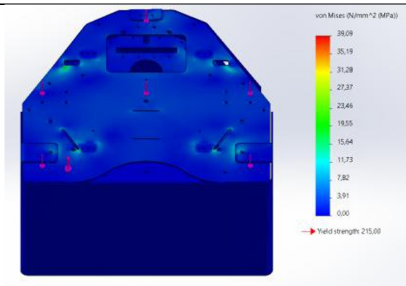

Fig. 26 Vertical load stress analysis results of 5052-H34

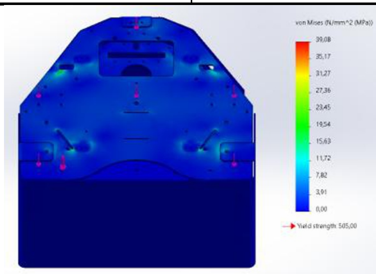

Fig. 27 Vertical load stress analysis results of 7075-T6

3.5 Displacement Analysis Vertical Load Force

Deformation due to vertical loads is shown in Figures 28–30. Aluminum 6061 experienced a displacement of 0.93604255mm, aluminum 5052-H34 experienced a displacement of 0.92265660mm, while aluminum 7075-T6 had the smallest displacement of 0.89700067mm. In general, the displacement values of the three materials were very small compared to the total dimensions of the frame, so they did not interfere with the stability of the robot.

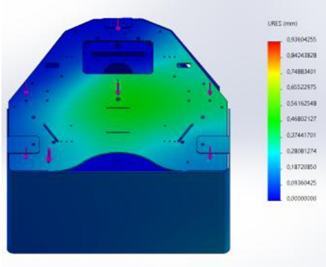

Fig. 28 Vertical load displacement analysis results of 6061

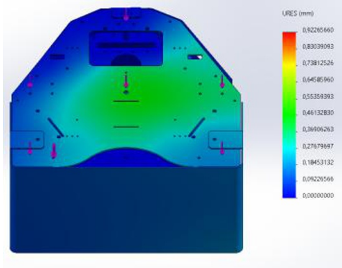

Fig. 29 Vertical load displacement analysis results of 5052-H34

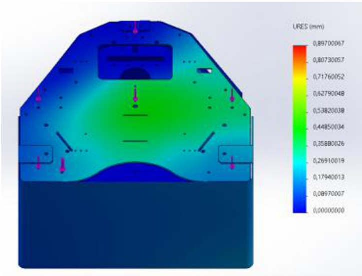

Fig. 30 Vertical load displacement analysis results of 7075-T6

3.6 Factor Of Safety Analysis Vertical Load Force

The safety factor (FOS) analysis under vertical loading shows that all three materials provide values greater than 1, which indicates that the frame is structurally safe under these conditions. For Aluminum 6061, the FOS is 1.41 Figure 31, meaning that despite its relatively low yield strength of about 55 MPa, the material can still withstand the applied vertical load. Aluminum 5052-H34 achieved an FOS of 5.50 Figure 32, offering a much larger margin of safety due to its higher yield strength of approximately 215 MPa. The highest FOS value 12.92, was obtained for Aluminum 7075-T6 Figure 33, confirming its superior resistance to vertical forces and demonstrating that it is the most reliable option among the tested materials.

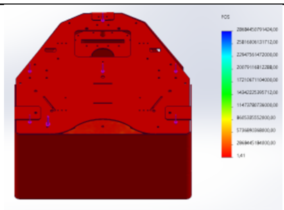

Fig. 31 Vertical load FOS analysis results of 6061

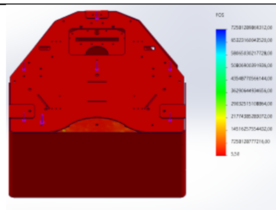

Fig. 32 Vertical load FOS analysis results of 5052-H34

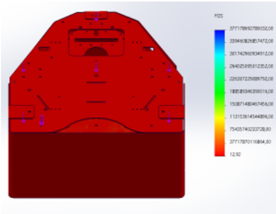

Fig. 33 Vertical load FOS analysis results of 7075-T6

3.7 Stress Analysis Ball Impact Load Force

The frame's maximum stress under ball impact loading, as shown in Figures 34–36, was 28.47 MPa for all three materials, according to the stress analysis. They have varying safety margins when compared to their respective yield strengths. Aluminum 6061's stress is getting close to its limit, indicating a possible risk of yielding under impact pressures (yield strength = 55 MPa). The biggest margin of safety is provided by Aluminum 7075-T6 (yield strength ≈ 505 MPa), while Aluminum 5052-H34 (yield strength = 215 MPa) stays within a safe range. Despite the fact that the applied impact stress is the same for all materials, these results show that the structural reliability is highly dependent on the mechanical characteristics of each alloy, with 7075-T6 showing the best resistance to ball impact loads.

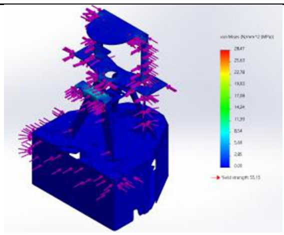

Fig. 34 Ball impact load stress analysis results of 6061

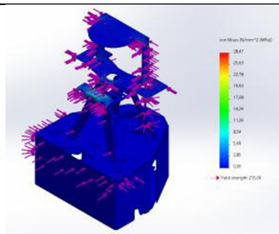

Fig. 35 Ball impact load stress analysis results of 5052-H34

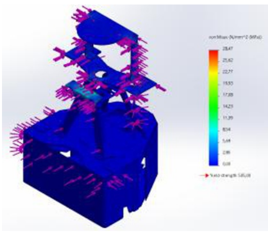

Fig. 36 Vertical load stress analysis results of 7075-T6

3.8 Displacement Analysis Ball Impact Load Force

The analysis of displacement under ball impact loading is displayed in Figures 37–39. For aluminum 6061, 5052-H34, and 7075-T6, the greatest displacements measured were 0.2579 mm, 0.2542 mm, and 0.2471 mm, respectively. These results show that the imposed ball load has little effect on the structural stability because they are so modest in relation to the robot frame's total size. However, in spite of having the highest displacement value, Aluminum 6061 is structurally unreliable because of its poor yield strength in relation to the impact stress, proving that a small displacement alone does not ensure safety.

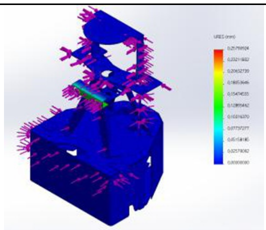

Fig. 37 Ball impact load displacement analysis results of 6061

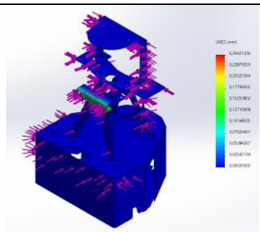

Fig. 38 Ball impact load displacement analysis results of 5052-H34

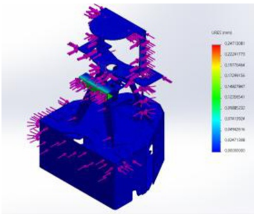

Fig. 39 Vertical load displacement analysisresults of 7075-T6

3.9 Factor Of Safety Analysis Ball Impact Load Force

The Factor of safety (FOS) analysis under ball impact loading is shown in Figures 40–42. The results show that all three materials are structurally safe since their FOS values are above 1. Aluminum 6061 has the lowest FOS of 1.94, meaning it can handle the applied load but with a small safety margin. Aluminum 5052-H34 has an FOS of 7.55, showing a much higher strength reserve and reliable performance. The highest FOS value, 17.74, was seen in Aluminum 7075-T6, proving it has the best resistance to impact forces and is the strongest option among the tested alloys.

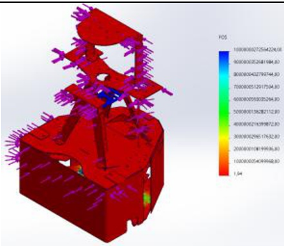

Fig. 40 Ball impact load displacement analysis results of 6061

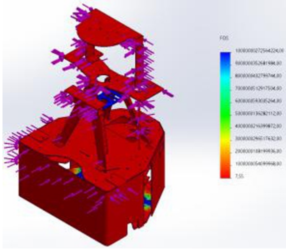

Fig. 41 Ball impact load displacment analysis results of 5052-H34

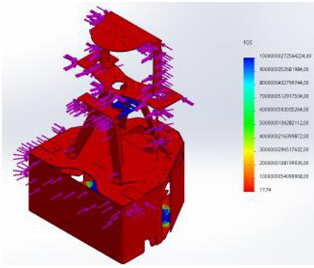

Fig. 42 Vertical load displacement analysisresults of 7075-T6

3.10 Validation Result

The results for horizontal, vertical, and ball impact loads are summarized in Tables 2–5. These tables present the maximum stress, displacement, and Factor of Safety (FOS) for each tested material under the corresponding loading conditions. For horizontal loading, both the front and back armor produced the same stress values of 0.1960 MPa and 0.4629 MPa across all materials. The identical stress arises because the applied load and frame geometry are constant, while the differences in FOS are governed by the yield strength of each alloy. In the vertical load scenario, stresses of approximately 39 MPa were observed, with Aluminum 6061 showing the lowest FOS of 1.41, indicating its limited margin of safety compared to Aluminum 5052-H34 (FOS = 5.50) and Aluminum 7075-T6 (FOS = 12.92). Under ball impact loading, the stress level was 28.47 MPa for all materials, but again the safety factors varied widely: 1.94 for 6061, 7.55 for 5052-H34, and 17.74 for 7075-T6. These results confirm that although stress values remain similar due to consistent load and geometry, the strength and reliability of the frame strongly depend on the yield strength of the selected material.

Material	Stress (MPa)	Displacement (mm)	Factor Of Safety
Aluminium 6061	0.1960	7.56e-06	281.35
Aluminium 5052-H34	0.1960	7.45e-06	1096.87

Aluminium 7075-T6	0.1960	724.e-06	2576.37

Table 2 Result of the horizontal load in front armor

Material	Stress (MPa)	Displacement (mm)	Factor Of Safety
Aluminium 6061	0.4629	1.626e-05	119.12
Aluminium 5052-H34	0.4629	1.603e-05	464.42
Aluminium 7075-T6	0.4629	1.558e-05	1090.84

Table 3 Result of the horizontal load in back armor

Material	Stress (MPa)	Displacement (mm)	Factor Of Safety
Aluminium 6061	39.09	0.93604255	1.41
Aluminium 5052-H34	39.09	0.92265660	5.50
Aluminium 7075-T6	39.08	0.89700067	12.92

Table 4 Result of the vertical load

Material	Stress (MPa)	Displacement (mm)	Factor Of Safety
Aluminium 6061	28.47	0.2579	1.94
Aluminium 5052-H34	28.47	0.2542	7.55
Aluminium 7075-T6	28.47	0.2471	17.74

Table 5 Result of the ball impact load

To validate these simulation results, a manual calculation was performed using Equation. 3, where the Factor of Safety is defined as the ratio of yield strength (σy) to maximum von Mises stress (σe). The manual computation for Aluminum 6061, with $\sigma y = 55.1485$ MPa and $\sigma e = 0.1960$ MPa, resulted in an FOS of 281.36, which matches closely with the SolidWorks output. Similar consistency was obtained for Aluminum 5052-H34 and Aluminum 7075-T6, demonstrating strong agreement between theoretical calculations and numerical simulations. This alignment confirms the reliability of the FEA method used in this study

`

$$Factor\ Of\ Safety = \frac{55.1485}{0.1960} = 281.36 \quad (3)$$

The result manual calculations yields very close agreement with the simulation outputs, proving the validity and reliability of the SolidWorks FEA results. This consistency guarantees that the stress and safety factor values derived from the simulation appropriately depict the structural behavior of the robot frame under the applied loads.

4 Conclusion

In engineering design, choosing the right material is vital to ensure that a structure can safely handle loads without excessive deformation or failure. This study assessed the structural performance of the Barelang63 wheeled goalkeeper robot frame under horizontal, vertical, and ball impact loads using Finite Element Analysis (FEA), comparing Aluminium 6061, Aluminium 5052-H34, and Aluminium 7075-T6. The results show that although stress and displacement values remain relatively low due to the large frame dimensions, the Factor of Safety (FOS) varies considerably based on the material properties. Under horizontal loads, the FOS values were 281.35, 1096.87, and 2576.37 in the front armor, and 119.12, 464.42, and 1090.84 in the back armor for Aluminium 6061, 5052-H34, and 7075-T6, respectively. For vertical loads, the FOS values were 1.41, 5.50, and 12.92, while for ball impacts they were 1.94, 7.55, and 17.74. These findings demonstrate that Aluminium 6061 has the lowest safety margin and thus provides limited resistance under vertical and impact loads. Aluminium 5052-H34 offers a good balance of strength, weight, and cost, making it a practical choice for competition. Meanwhile, Aluminium 7075-T6 displays the highest strength and minimal deformation, making it the most dependable option in terms of durability and structural safety. Future research should include experimental validation, fatigue testing, and dynamic load simulations to confirm these numerical findings and to explore the potential of hybrid or composite materials for further performance enhancement.

References

1. M. Asada et al., “Middle Size Robot League Rules and Regulations for 2024,” 2024.
2. Balai Pengembangan Talenta Indonesia Pusat Prestasi Nasional Kementerian Pendidikan Dan Teknologi, “BUKU PEDOMAN KONTES ROBOT INDONESIA (KRI) TAHUN 2023,” Feb. 2023.
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, pp. 8–17, Nov. 2023
4. A. Wibisana, M. Imron Shodiq, M. Naufal Airlangga Diputra, and N. Nazhifah, “Performance Comparison of Wheeled Soccer Robot Frames Using Finite Element Analysis (FEA),” JURNAL INTEGRASI, vol. 17, no. 1, pp. 59–69, Apr. 2025
5. A. Welch-Phillips, D. Gibbons, D. P. Ahern, and J. S. Butler, “What Is Finite Element Analysis?,” Clinical Spine Surgery: A Spine Publication, vol. 33, no. 8, pp. 323–324, Jul. 2020.
6. M. Fazly Arliansyah, Saputra, Rahman, W. Suyudi Ashari, A. Dasira, and M. Zaki Prawira, “ANALISA FINITE ELEMENT METHOD (FEM) UJI BEBAN PADA MEJA POLYETHYLENE,” Analisa Finite Element Method (FEM) Uji Beban Pada Meja Polyethylene, vol. 4, no. 2, 2023.
7. Iqbal Dumyati and Soffan Nurhaji, “Modeling dan Simulasi Finite Element Analysis pada Segitiga T Sepeda Motor Menggunakan Sofware Ansys 2023,” Quantum Teknika, vol. 5, no. 1, pp. 26–30, Nov. 2023.

8. I. Muhlisin and S. Sudiman, "Pengaruh Variasi Beban terhadap Faktor Kekuatan Rangka Sepeda dari Bahan AISI 1035 Steel (SS) dengan Simulasi Solidworks," Briliant: Jurnal Riset dan Konseptual, vol. 9, no. 1, p. 236, Feb. 2024
9. M. Y. Wibowo, Indra Maulana, Andi Abdullah Ghyferi, B. A. Kurniawan, and Miftahun Nuril, "Perancangan Chassis Prototype Mobil Warak dan Simulasi Statik dengan Metode Finite Element Analysis," Jurnal Mekanik Terapan, vol. 3, no. 3, pp. 86–92, Dec. 2022.
10. None Sandy Suryady and None Eko Aprianto Nugroho, "SIMULASI FAKTOR KEAMANADAN PEMBEBANAN STATIK RANGKA PADA TURBIN ANGIN SAVONIUS," Jurnal Ilmiah Multidisiplin, vol. 1, no. 2, pp. 42–48, Mar. 2022.
11. F. A. Budiman, A. Septiyanto, S. Sudiyono, A. D. N. I. Musyono, and R. Setiadi, "Analisis Tegangan von Mises dan Safety Factor pada Chassis Kendaraan Listrik Tipe In-Wheel," Jurnal Rekayasa Mesin, vol. 16, no. 1, p. 100, Apr. 2021.
12. R. Ardilla, W. Haganta Ginting, and R. Cornelius Siagian, "Pengaruh Latihan Otot Tungkai Tanpa Alat Terhadap Kekuatan Tendangan," Pengaruh Latihan Otot Tungkai Tanpa Alat Terhadap Kekuatan Tendangan, vol. 4, 2023.
13. Abdullah Hossain Al Newaz. "“Structural Analysis and Design of a Curved Anchor Bracket Using SolidWorks.”" *International Journal of Scientific and Research Publications*, vol. 15, no. 1, 2025. Accessed 11 Oct. 2025.
14. Muhammad Amin Sabtu, et al. "Finite Element Analysis of Stress Distribution in AL6061 Frame Structures under Varying Applied Loads." International Journal of Integrated Engineering, vol. 16, no. 2, 16 Apr. 2024. Accessed 25 Nov. 2024.
15. P. Kurowski, Engineering Analysis with SolidWorks Simulation 2019. 2019.
16. N. Agus, K. R. Dantes, and N. Pasek, "ANALISIS TEGANGAN STATIK PADA RANCANGAN FRAME MOBIL LISTRIK GANESHA SAKTI (GASKI) MENGGUNAKAN SOFTWARE SOLIDWORKS 2014," Jurnal Pendidikan Teknik Mesin Undiksha, vol. 6, no. 2, pp. 113–113, Aug. 2018.