OPTIMIZATION OF OMNI-WHEELED ROBOT BASE DESIGN WITH VARIATIONS IN WHEEL CONFIGURATION AND FRAME PROFILE USING FINITE ELEMENT METHOD

Abdi Muhammad[1)], Ella Sundari[2)*]
[1)]Mechanical Engineering Production and Maintenance Study Program, Mechanical Engineering Department, Politeknik Negeri Sriwijaya
Jl. Srijaya Negara Bukit Besar Palembang 30139, Indonesia
[2)] Mechanical Engineering Department, Politeknik Negeri Sriwijaya
Jl. Srijaya Negara Bukit Besar Palembang 30139, Indonesia
[*] Corresponding email: e_sundari@polsri.ac.id

ARTICLE INFORMATION

Revised
23/11/2025

Accepted
01/12/2025

Online Publication
03/06/2026

©2026 The Authors. Published by AUSTENIT (Indexed in SINTA)

doi:

10.36257/austenit.v18i1.11336

ABSTRACT

This research focuses on optimizing the design of an omni-wheeled robot base by exploring variations in wheel arrangements and frame profiles through the Finite Element Method. Three-wheel configurations were examined: the X-Drive configuration (four wheels at a 45° angle), a four-wheel side arrangement, and a circular four-wheel setup. Each configuration was assessed using two different frame materials aluminum hollow 6061 and aluminum extrusion profile 6063. FEM simulations were carried out to evaluate von mises stress, total deformation, and safety factor for each model. The findings reveal that the aluminum hollow 6061 frame outperforms the aluminum extrusion profile 6063, showing lower stress values, minimal deformation, and higher safety factors. The M2 configuration (four wheels on the sides) with aluminum hollow 6061 recorded the lowest stress (14.163 MPa) and the highest safety factor (15). Meanwhile, the smallest deformation (0.01 mm) was observed in the M1 design for both aluminum hollow 6061 and aluminum extrusion profile 6063 materials. The outcomes of this research present an optimal design recommendation applicable to KRAI robots as well as other types of omni-wheeled robots.

Keywords *: Autodesk Fusion 360, Based Design, Finite element metho Mesh Convergence, Omni-wheeled robot*

1 INTRODUCTION

The development of robotics technology in Indonesia in recent years has shown quite rapid progress, particularly through competitions such as the Indonesian ABU Robot Contest (KRAI), which is part of the Indonesian Robot Contest (KRI). One of the main components commonly used in KRAI robots is omni-wheels, which enable holonomic movement—that is, the ability to move in any direction without needing to change the robot's primary orientation. Omni-directional robot platforms offer mobility advantages over conventional designs, especially when operating in crowded environments (Amarasiri et al., 2022).

The growing interest in robotics research has significantly driven scientific developments in this field (Jatmiko et al., 2012). In recent years, demand for flexible production systems has increased, particularly those capable of producing personalized and tailored products to various needs. For example, the German government refers to these flexible production systems as “smart factories” and is promoting the industry 4.0 initiative to achieve this (Rojko, 2017).

The use of omni wheels provides advantages over conventional wheels, especially in terms of flexibility of movement and navigation capabilities in dynamic environments.

An omni wheeled robot's ability to move freely depends heavily on the type of platform used (Nur & Indrabulan, 2021). An optimal base design must be able to support loads, absorb forces generated during movement, and minimize deformation to prevent stability and movement accuracy.

The superiority of omni-wheels has been proven not only in domestic competitions such as KRAI, but also in international competitions such as Robocup, Vex Robotics, and ABU Robocon. Furthermore, these wheels are also widely applied in industry, such as in logistics robots in automated warehouses and AGVs (Automated Guided

Vehicles). Their ability to maneuver freely in confined spaces makes them an ideal choice for various industrial applications (Amarasiri et al., 2022). In real-world applications, a strong and deformation-resistant robot base structure is crucial for maintaining operational reliability and safety.

According to Sani et. al (2024), the use of the FEA method can reduce costs and time, and can be used before the actual prototype is made.

Based on these needs, this research aims to design and analyze various omni robot base configurations using the finite element method (FEM). The three main configurations to be studied include: (1) four omni wheels at a 45° angle, (2) four wheels on the side of the base, and (3) four wheels arranged in a circle at the corner of the base. Each configuration will have two variations of the frame shape, so that a total of six designs will be analyzed based on the von Mises stress parameters, total deformation, and safety factor. The results of the study are expected to provide information on the mechanical performance of each design as well as recommendations for the best design to be applied both in the world of robotics competitions and real industry.

2. MATERIALS AND METHODS

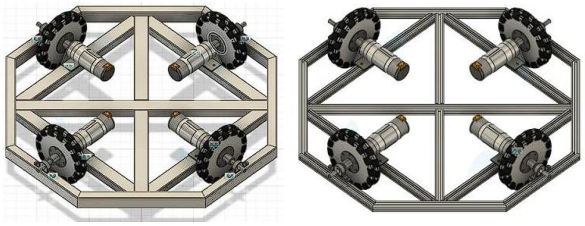
2.1. Research Flow

This research process begins with an initial stage of observation and literature review to understand the problem and relevant technical references. After that, a basic robot design was created using CAD software in Autodesk Fusion 360. The design data was then input into a Finite Element Method (FEM)-based simulation module, followed by a mesh generation process to divide the model into small elements to be analyze numerically.

The next stage is the simulation or solving process, which includes analysis of von Mises stress, deformation, and safety factor. The simulation results are used to evaluate whether the construction design is safe. If the design is declared unsafe, revisions are necessary, and the process returns to the previous stage. If the design is deemed safe, a mesh convergence test is performed to ensure that the simulation results do not change significantly with variations in mesh size. If the calculation results have not converged, the meshing and simulation processes are repeated. However, if the results have converged, the simulation data will be processed and analyzed further. The end of this process is drawing conclusions and providing recommendations based on the results obtained.

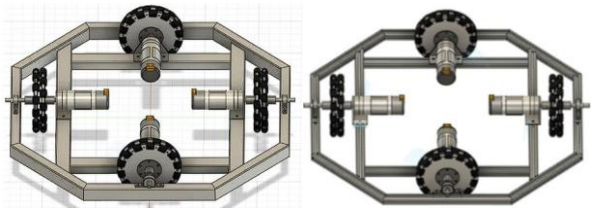
2.2. Design

Three models were created, each with two frame variations, for a total of six designs. The base design, which uses a 45° configuration, is named the M1 Octagon Cross as can be seen in Figure 1. The base design with wheels on the side is named the M2 Octagon Square which can be seen in Figure 2. And the last design with circular wheels is named the M3 Octagon Inline as in Figure 3.



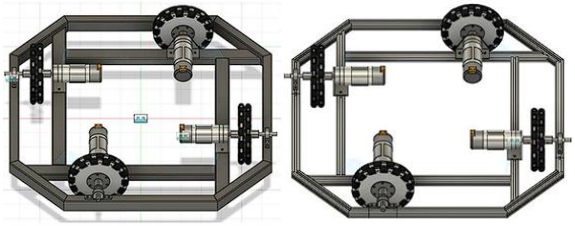
(a) (b)

Figure 1. M1 Design (a) Al 6061 (b) Al 6063



(a) *(b)*

Figure 2. M2 Design (a) Al 6061 (b) Al 6063



(a) *(b)*

Figure 3. M3 Design (a) Al 6061 (b) Al 6063

2.3. Material

This research uses 2 materials with different specifications, Aluminum 6061, and Aluminum 6063. In Table 1 are the specifications of each material.

Table 1. Material Properties of Aluminum 6061 and Aluminum 6063 (Seli et al., 2013).

Material Properties	Aluminum 6061	Aluminum 6063
Young's Modulus	69 GPa	69 GPa
Poisson's ratio	0.33	0.33
Shear Modulus	25.8 GPa	25.8 GPa
Density	2.7 g/cc	2.7 g/cc
Yield Strength	277 MPa	145 MPa
Tensile Strength	310 MPa	186 MPa

2.4. Mesh

Deep meshing is the process of forming a network of small elements that represent the overall geometric model (Nur Arif et al., 2024). The process of dividing an object into several small elements is known as meshing (Sani et al., 2024). Table 2 shows the number of meshes for each design.

Table 2. Number of Mesh and Nodes

Design	Material	Mesh	Nodes
M1	Al 6061	613.802	1.222.506
	Al 6063	392.974	758.888
M2	Al 6061	494.251	977.375
	Al 6063	301.085	572.225
M3	Al 6061	514.147	1.013.408
	Al 6063	327.067	629.355

3. RESULTS AND DISCUSSION

The finite element method is a numerical solution that can be used to obtain solutions to complex or impossible to calculate problems (Tjong, 2015). FEM is a numerical method used to obtain approximate solutions to complex engineering problems (Suratkar et al., 2013). In this study, a load of 500 N as applied during the simulation. The value of 500 N was chosen based on the load regulations of the Indonesian ABU Robot Contest (KRAI), where the maximum allowable robot load is 50 kg The results of simulation values for each material and geometry can be seen in Table 3

Table 3. Test Results

FEM Test Results					
No	Design	Material	Stress (MPa)	Deformation (mm)	Safety factor
1	M1	Al 6061	19.480	0.010	14.11
		Al 6063	65.443	0.010	04.43
2	M2	Al 6061	14.163	0.013	15.00
		Al 6063	37.496	0.063	03.86
3	M3	Al 6061	18.183	0.020	15.00
		Al 6063	41.537	0.114	03.49

Details for each simulation result on the 6 designs can be seen in figures 4 to 21.

3.1. Von Misses Stress

Von Mises stress is a stress parameter used to predict the potential failure of a material when subjected to static loads. Von Mises stress is a very useful criterion in engineering design, especially for predicting material failure due to loading and this criterion can be used for materials experiencing combined stresses (Juvinall, 2012).

The details of the von Mises stress results by using Autodesk Fusion 360 simulations on each design can be seen in Figure 4 to 9 below.

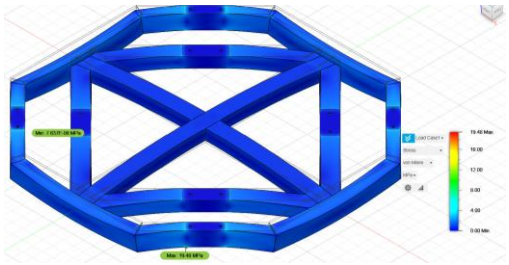


Figure 4. M1 Base Al 6061 Von mises stress

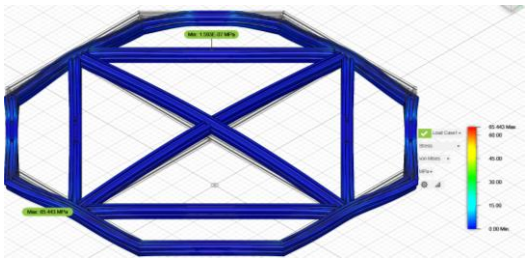


Figure 5. M1 Base Al 6063 Von mises stress

In M1 Design with Al 6061 material, the maximum von mises stress value was 19.48 MPa and the minimum was $7.637x10^{-8}$ MPa, while in Al 6063 material, the maximum von mises stress value was 65.443 MPa and the minimum value was $1.593x10^{-7}$ MPa.

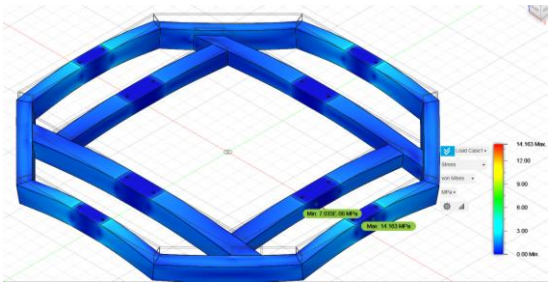


Figure 6. M2 Base Al 6061 Von mises stress

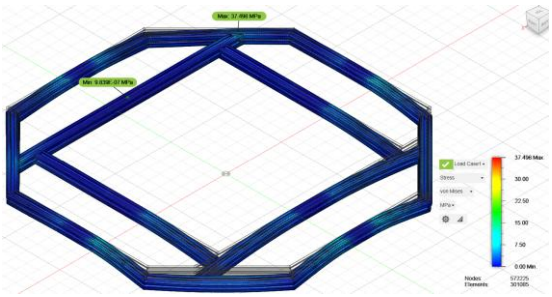


Figure 7. M2 Base Al 6063 Von mises stress

In M2 Design with Al 6061 material, the maximum von mises stress value was 14.163 MPa and the minimum value was $7.035x10^{-8}$ MPa, while in Al 6063 material, the maximum von mises stress value was 37.496 MPa and the minimum was $9.839x10^{-7}$ MPa.

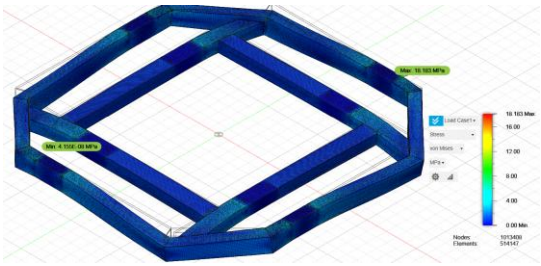


Figure 8. M3 Base Al 6061 Von mises stress

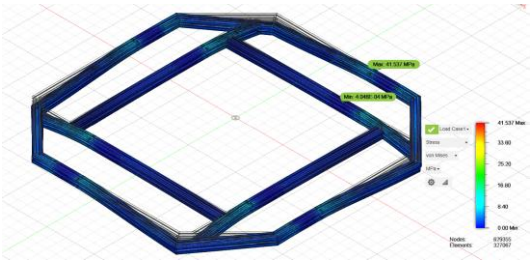


Figure 9. M3 Base Al 6063 Von mises stress

In M3 Design with Al 6061 material, the maximum von mises stress value was 18.183 MPa and the minimum was 4.155×10^{-8} MPa, while in Al 6063 material, the maximum von mises stress value was 41.537 MPa and the minimum was 4.046×10^{-8} MPa.

The performance differences between Aluminum 6061 and Aluminum 6063, which range from 2 to 3 times in terms of stress distribution, occur due to the disparity in their mechanical properties. Aluminum 6061 has a yield strength nearly twice that of Aluminum 6063. Since the FEM simulation evaluates structural response based on applied loads, geometry, and material strength, a material with a higher yield strength naturally exhibits lower stress values, smaller deformation, and significantly higher safety factors. Therefore, the 2–3 times difference in performance directly reflects the inherent difference in mechanical strength between the two materials.

3.2. Deformation

Deformation is the change in shape of a material due to an applied force. During the deformation process, the material absorbs energy due to the force applied to it. Based on its nature, deformation is divided into two types: elastic and plastic. Elastic deformation occurs when the material undergoes a temporary change in shape and returns to its original shape after the load is removed. In contrast, plastic deformation is permanent and persists even after the force is removed. In this analysis, the observed deformation is the total deformation of the robot's base frame when subjected to loading, which provides an indication of the extent of geometric change that occurs during operation (Dionysius et al., 2023).

The details of the deformation results by using Autodesk Fusion 360 simulations on each design can be seen in Figure 10 to 15 below.

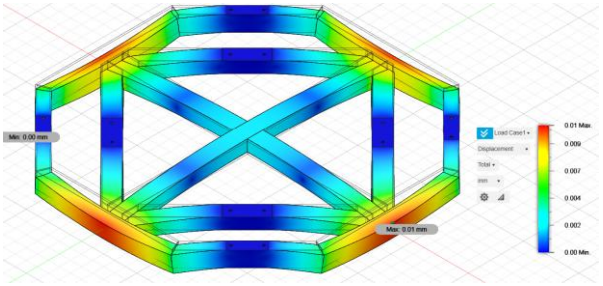


Figure 10. M1 Base Al 6061 Deformation

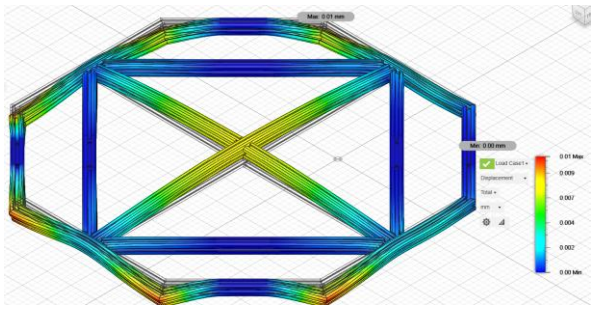


Figure 11. M1 Base Al 6063 Deformation

In M1 Design with Al 6061 material, the deformation value was 0.01 mm and in Al 6063 material it was 0.01 mm.

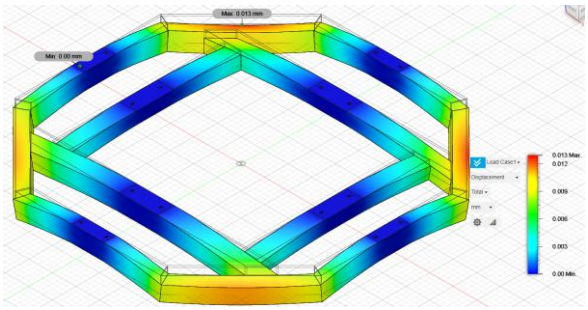


Figure 12. M2 Base Al 6061 Deformation

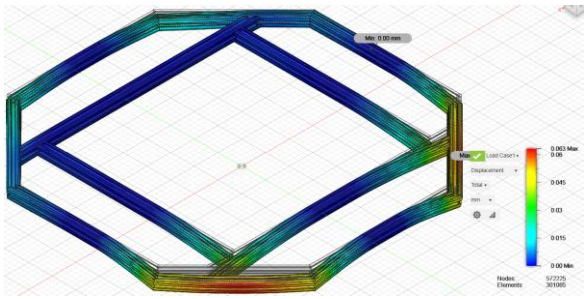


Figure 13. M2 Base Al 6063 Deformation

In M2 Design with Al 6061 material, the deformation value was 0.013 mm and in Al 6063 material it was 0.063 mm.

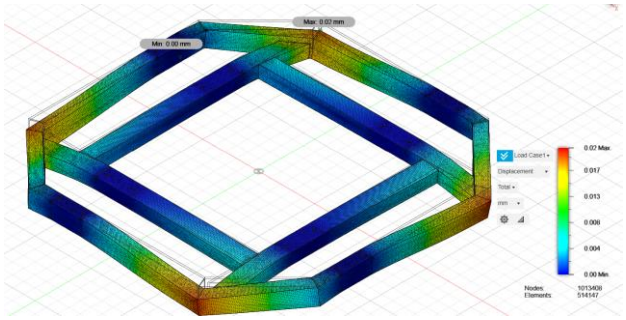


Figure 14. M3 Base Al 6061 Deformation

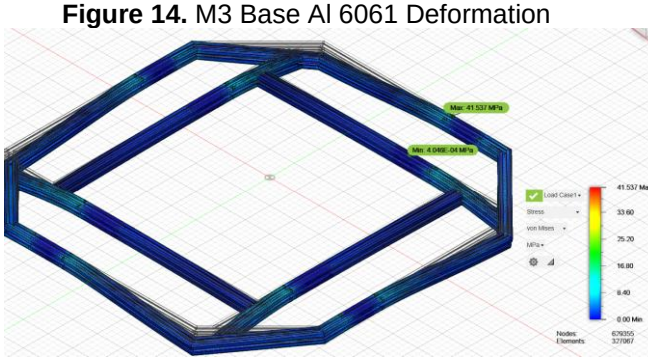


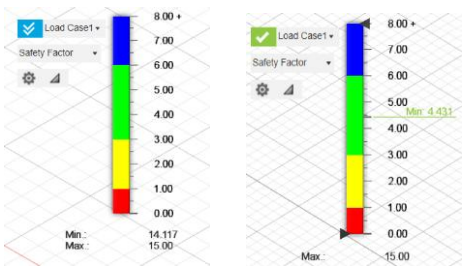
Figure 15. M3 Base Al 6063 Deformation

In M3 Design with Al 6061 material, the deformation value was 0.02 mm and in Al 6063 material it was 0.114 mm.

The deformation difference between Aluminum 6061 and Aluminum 6063 is not as large as the difference in stress because both materials have the same Young's Modulus (69 GPa). In linear static FEM, deformation is determined mainly by stiffness, and stiffness is directly proportional to Young's Modulus. Since both materials share the same modulus, their resistance to elastic deflection is almost identical.

3.3. Safety Factor

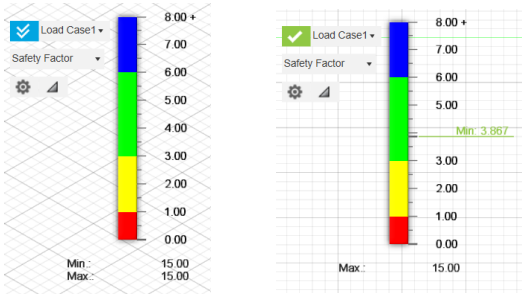
In figure 16, 17 and 18, can be seen the safety factor as the ratio between the strength of a material and the maximum applied load. The safety factor is used as a basis for evaluation to ensure that machine component designs remain safe even at their minimum dimensions (Khaitami et al., 2022). This factor is used to ensure that a structure or component remains safe despite variations in material, manufacturing, or operational conditions.



(a) (b)

Figure 16. M1 Base Safety factor (a) Al 6061 and (b) Al 6063

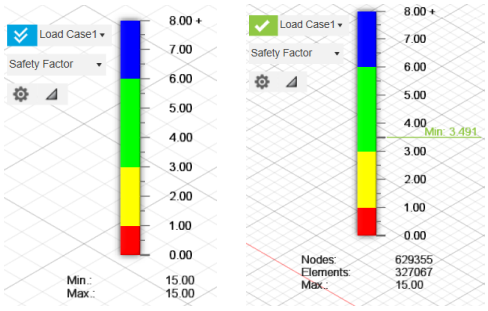
In M1 Design with Al 6061 material, the safety factor value was 14.11, while in Al 6063 material, the value was 4.431.



(a) (b)

Figure 17. M2 Base Safety factor (a) Al 6061 and (b) Al 6063

In M2 Design with Al 6061 material, the safety factor value was 15, while in Al 6063 material, the value was 3.867.



(a) (b)

Figure 18. M3 Base Safety factor (a) Al 6061 and (b) Al 6063

In the M3 design with Al 6061 material, the safety factor value was 15, while in the Al 6063 material, the value was 3.491.

3.4. Mesh Convergence

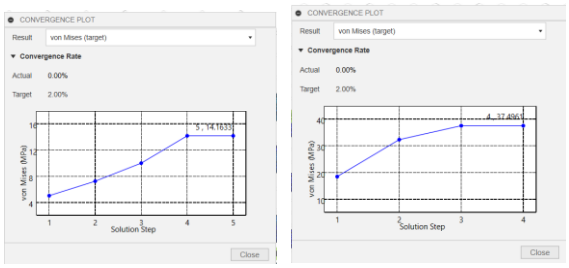
The mesh convergence test is a procedure to ensure that the results of an FEM analysis are not affected by the size of the mesh elements used. The convergence test is performed to determine the best size of the elements, considering the computational load and accuracy of the results (Owen F. Hughes and Jeom Kee Paik, 2010).

The convergence test is considered successful if the calculation error due to changes in the mesh structure is less than 2% (Putra & Misbah, 2022). Convergence is expected at this stage; if convergence is not achieved, the process must be repeated by correcting the mesh. Conversely, if the results are convergent, the process can be proceed to the next stage (Saputra & Romahadi, 2023). Figure 19 to 21 shows a mesh convergence graph for each test on each design and material.



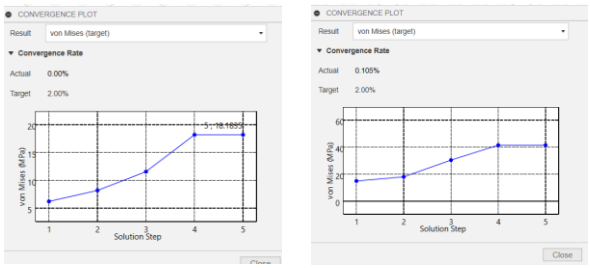
(a) (b)

Figure 19. M1 Base Mesh Convergence Graph (a) Al 6061 and (b) Al 6063



(a) (b)

Figure 20. M2 Base Mesh Convergence Graph (a) Al 6061 and (b) Al 6063



(a) (b)

Figure 21. M3 Base Mesh Convergence Graph (a) Al 6061 and (b) Al 6063

4. CONCLUSION

The conclusions that can be drawn from the simulation results for the six designs are:

1. Based on the simulation results for von Mises stress, deformation, and safety factor, it is known that the base design using 6061 hollow aluminum material has superior performance compared to 6063 extruded aluminum material.
2. After obtaining the von Mises stress values for the six designs, it was determined that M2 design, using 6061 hollow aluminum material, had the lowest stress value. Meanwhile, M1 design had the lowest deformation value for both materials. For safety factor, all designs had yield strength values above the load, but the highest safety factor value was found in M2 design using 6061 hollow aluminum material.
3. Based on the mesh convergence test graph, it was found that changes in the simulation results occurred when the mesh size was changed, and the results were taken after no change or convergence occurred.

REFERENCES

Amarasiri, N., Barhorst, A. A., & Gottumukkala, R. (2022). Robust Dynamic Modeling and Trajectory Tracking Controller of a Universal *Omni*-Wheeled Mobile Robot. *ASME Letters in Dynamic Systems and Control*, *2*(4). https://doi.org/10.1115/1.4055690

Diinil Mustaqiem, A., Nurato. (2020). Analisis Perbandingan Faktor Keamanan Rangka Scooter. *Jurnal Teknik Mesin*, *09* (3), 164–172. http://dx.doi.org/10.22441/jtm.v9i3.9567

Dionysius, A. K., Widodo, A., & Kurdi, O. (2023). Analisis Deformasi Dan Tegangan Pada Friction Block Kereta Api Cepat Dengan Bentuk Heptagon Menggunakan Finite Element Method. *Jurnal Teknik Mesin S-1*, *11*(4), 215–218. https://ejournal3.undip.ac.id/index.php/jtm/article/view/41869

Jatmiko, W., Mursanto, P., Tawakal, M. I., Alvissalim, M. S., Hafidh, A., Budianto, E., Kurniawan, M. N., Ahfa, K., Danniswara, K., Ma'sum, M. A., Hermawan, I., & Jati, G. (2012). *Robotika Teori Dan Aplikasi*.

Juvinall, R. C. and K. M. M. (2012). *Fundamentals of Machine Component Design Fifth Edition*. John Wiley & Sons, Inc.

Khaitami., K. M. A., Arifin, F., Sundari. E., (2022). Desain Alat Bantu Troli Penggulung Selang Pemadam Kebakaran Semi Otomatis. *Machinery Jurnal Teknologi Terapan*, *3*(1), 29–36. https://jurnal.polsri.ac.id/index.php/machinery/article/view/4472/1889

Nur Arif, I., Nugroho, S., & Jamari. (2024). Studi Pendahuluan Rekonstruksi Dan *Mesh* Tulang Lumbar 1 Sampai 2 Menggunakan Metode Finite Element. In *Jurnal Teknik Mesin S-1* (Vol. 12, Issue 2). https://ejournal3.undip.ac.id/index.php/jtm/article/view/44852

Nur, M., & Indrabulan, T. (2021). Rancang Bangun Smart Robot *Omni* Wheel sebagai Permainan Edukatif dan Interaktif bagi Balita. *PROtek : Jurnal Ilmiah Teknik Elektro*, *8*(1), 45–48. https://doi.org/10.33387/protk.v8i1.2520

Owen F. Hughes and Jeom Kee Paik. (2010). Structural Analysis and Design. *Examples in Structural Analysis*, 17–38. https://doi.org/10.1201/b16336-5

Putra, J. A., & Misbah, M. N. (2022). Studi Pengaruh Ukuran Bracket Pondasi Mesin terhadap Tegangan dengan Menggunakan Finite Element Method. *Jurnal Teknik ITS*, *11*(1), 1–6. https://doi.org/10.12962/j23373539.v11i1.82025

Rojko, A. (2017). Industry 4.0 concept: Background and overview. *International Journal of Interactive Mobile Technologies*, *11*(5), 77–90. https://doi.org/10.3991/ijim.v11i5.7072

Sani, A. A., Gunawan, I., Fernandes, Y. E., Suryana, D., & Nugraha, A. (2024). Optimalisasi Design Sambungan Aluminium Extrusion 2020 Menggunakan Metode Finite Element Analysis (Fea). *Austenit*, *16*(1), 9–14. https://doi.org/10.53893/austenit.v16i1.7572

Saputra, G. J., & Romahadi, D. (2023). Analisis Komparasi Kekuatan Geometri Desain Rusuk Penguat pada Kursi Plastik menggunakan Computer Aided Engineering. *Jurnal Teknik Mesin*, *12*(1), 20. https://doi.org/10.22441/jtm.v12i1.16503

Seli, H., Awang, M., Ismail, A. I. M., Rachman, E., & Ahmad, Z. A. (2013). Evaluation of properties and FEM model of the friction welded mild steel-Al6061-alumina. *Materials Research*, *16*(2), 453–467. https://doi.org/10.1590/S151614392012005000178

Suratkar, A., Shukla, V., & Zakiuddin, K. S. (2013). Design Optimization Of Overhead EOT Crane Box Girder Using Finite Element Analysis. *International Journal of Engineering Research & Technology (IJERT)*, *2*(7), 720–724. https://www.ijert.org/research/design-optimization-of-overhead-eot-crane-box-girder-using-finite-element-analysis-IJERTV2IS70304.pdf

Tjong, W. F. (2015). *Pengantar Metode Elemen Hingga Untuk Analisis Struktur*. Rajawali Pers.