



Improvement of the aerodynamic performance of the Mataram University electric vehicle prototype

S. Suryadi, H.S. Tira*, I.G.N.K. Yudhyadi

Mechanical Engineering Department, Engineering Faculty, the University of Mataram, Jl. Majapahit no. 62, Mataram, NTB, 83125, Indonesia. HP. +6287878580219

*E-mail: hendrytira@unram.ac.id

ARTICLE INFO

ABSTRACT

Article History:

Received 24-10-2025

Accepted 07-02-2026

Available online 01-04-2026

Keywords:

Drag coefficient

Drag force

Prototype vehicle

Energy efficiency

Computational fluid dynamics



The number of motor vehicles worldwide increases every year, leading to higher pollution levels. To address this issue, various efforts have been made, one of which is improving vehicle design to enhance aerodynamics. This study aims to analyze the improvement in aerodynamic performance of the Mataram University prototype electric vehicle design. The computational fluid dynamics method was employed to achieve this objective. In this study, velocity variations of 10.48 m/s, 14.65 m/s, 18.82 m/s, 22.98 m/s, and 27.15 m/s were used. The results indicate an improvement in aerodynamics, as evidenced by a 5.682% reduction in the average drag coefficient compared to the initial design (2023). This reduction in drag coefficient signifies an enhancement in the vehicle's aerodynamic efficiency, which positively impacts overall efficiency and reduces energy consumption.

Dinamika Teknik Mesin, Vol. 16, No. 1, April 2026, p. ISSN: 2088-088X, e. ISSN: 2502-1729

1. INTRODUCTION

The increasing population also leads to a growing demand for energy, particularly fossil fuels. However, these energy sources are non-renewable, causing their availability to decline over time, Setyono and Kiono (2021). This condition contrasts with the continuously rising number of vehicles. In 2021, the number of registered vehicles reached 141,992,573 units, and by 2022, it had increased to 148,261,817 units, Badan Pusat Statistik (2024). The growing number of motor vehicles has negative impacts, as exhaust gas emissions contribute to global warming, which can harm human health and the surrounding environment (Rahmadania, 2022; Kurnia, 2021). One of the efforts to reduce fuel consumption is by designing energy-efficient vehicles, Tira et al. (2023).

The efficiency of a vehicle can be determined through its level of aerodynamics, Hanafi et al. (2024). Aerodynamics is the study of forces caused by the relative motion of a fluid over a surface, Dwi et al. (2021). From an aerodynamic perspective, it is closely related to the streamline shape of the vehicle, which is designed according to the characteristics of the airflow impacting the vehicle's surface, thereby reducing the resistance experienced while in motion, Rois and Dzulfikar (2021). The methods used to determine the aerodynamic shape of a vehicle include experiments conducted in a wind tunnel or simulations using CFD (Computational Fluid Dynamics) software, Prastyo et al. (2020).

The forces that influence the aerodynamic performance of a vehicle are the drag force and the lift force, Ananda et al. (2021). The drag force is the resistance acting on an object moving through a fluid, which acts parallel but in the opposite direction to the object's velocity vector. The drag coefficient represents the magnitude of the fluid resistance acting on the object. Meanwhile, the lift force is the upward force that causes an object to rise due to the pressure beneath the object being greater than the pressure above it. The lift coefficient indicates the magnitude of the lift force acting on the object, Nurcahyo and Wahyudi (2021).

The drag coefficient plays a crucial role in vehicle efficiency. A more aerodynamic vehicle body will result in a lower drag coefficient value, thereby enhancing engine power optimization and reducing fuel consumption due to decreased air resistance, Hanif and Utomo (2015). Several factors can influence the drag coefficient value, including body shape, drag force (FD), fluid velocity, fluid density, and the cross-sectional area perpendicular to the direction of the fluid flow, Suryady and Zhafran (2022).

Based on the research conducted by Aprillia and Sriyono (2023) on the aerodynamic analysis of an urban single-seater vehicle body using the CFD (Computational Fluid Dynamics) method, which compared two designs, design 1 and design 2 under varying speeds of 8.3 m/s, 11.1 m/s, and 13.9 m/s, it was found that the lowest drag coefficient (Cd) value occurred in design 2, with a Cd of 0.29 at a speed of 13.9 m/s. At the same speed, the drag coefficient for design 1 was 0.38. Based on these results, it can be concluded that the body design shape and speed variation significantly influence the vehicle's drag coefficient value.

Based on the explanation above, the objective of this study is to determine the improvement in the aerodynamic performance of the Mataram University prototype vehicle body design from 2023 to 2024 using the CFD (Computational Fluid Dynamics) method with varied vehicle speeds, analyzed through ANSYS software. The significant difference between the 2024 and 2023 body designs lies in the slimmer and more compact profile of the 2024 model. With a more streamlined shape and a reduced frontal area, the design is optimized to achieve superior aerodynamic efficiency. Mataram University has participated in the Shell Eco-Marathon competition twice, in 2023 and 2024. The Shell Eco-Marathon is an event aimed at developing future vehicles with high efficiency and low energy consumption, Wibowo et al. (2022). Therefore, this simulation was conducted to identify the extent of aerodynamic improvement achieved through the redesigned vehicle body.

2. RESEARCH METHODS

In this study, data collection was carried out using the Computational Fluid Dynamics (CFD) simulation method with Ansys 2024R1 software. The vehicle body design was created using Fusion 360 software. Figure 1 shows the differences in the body design of the Universitas Mataram prototype vehicle between the 2023 and 2024 models, while the vehicle dimensions are presented in table 1.

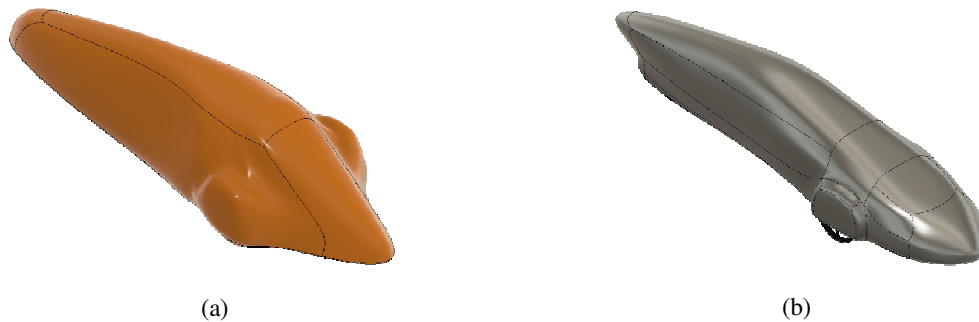


Figure 1. Vehicle designs used in the simulation: (a) 2023 vehicle design, (b) 2024 vehicle design.

Table 1. Vehicle dimensions

Dimensions	2023 design data	2024 design data
Length	3570 mm	3274 mm
Width	1213 mm	788 mm
Height	829 mm	752 mm
Front angle of the car	30°	17°
Frontal area	0.6872 m ²	0.4 m ²

The variables in this study are divided into two categories: dependent variables and independent variables. Dependent variables are variables that are influenced by independent variables, which in this study are drag force (FD) and drag coefficient (Cd). Meanwhile, independent variables are variables that influence the results of the study and can be varied. The independent variables in this study are variations in the design of the

prototype vehicle body, namely the 2023 body design and the 2024 body design. In addition, there are also variations in air speed, namely 10.48 m/s, 14.65 m/s, 18.82 m/s, 22.98 m/s, and 27.15 m/s, which include the average wind speed in the West Nusa Tenggara region of 2.15 m/s, Andriyani and Munastha (2022).

The working principle of this simulation is to place a car in a simulation domain and set the air velocity as desired, then obtain the drag force and drag coefficient values from the simulation. Before the simulation is carried out, a simulation domain is first created, as shown in figure 2. The simulation domain used is 24.5 m long, 2 m wide, and 7 m high, Amaluddin (2022). Then, the meshing stage is carried out. In this simulation, the type of mesh used is poly-hexcore, which has high accuracy even though the number of meshes is small, thus reducing simulation time, Widiarta et al. (2022). Within the simulation domain, there is a body of influent that has a finer mesh size in order to obtain more precise simulation results. The results of the mesh can be seen in figure 3. In addition, several parameters are used in the simulation, which are shown in table 2.

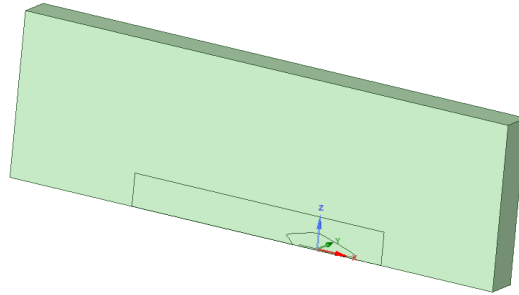


Figure 2. Simulation domain

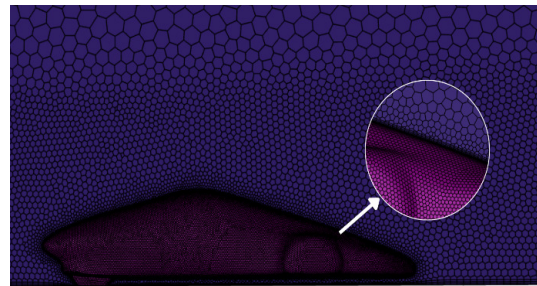


Figure 3. Meshing process

Table 2. Parameters in the simulation

Simulation parameters	Selected values/models
Turbulence model	K- ω SST (<i>shear stress transport</i>)
Type of fluid	Air
Temperature	300 K
Pressure	101325 Pa
Density	1.17662 kg/m ³
Viscosity	1.846 $\times 10^{-5}$ kg/m.s
Inlet air speed	10.48 m/s, 14.65 m/s, 18.82 m/s, 22.98 m/s, 27.15 m/s

3. RESULTS AND DISCUSSION

3.1 Mesh validation

The meshing process in this study aims to enable calculations using simulation methods. Therefore, the quality of the mesh greatly affects the results of the computation. In analyzing the quality of the mesh in Ansys software, two standards are used, namely skewness and orthogonal quality, Firmanto (2023). The standards for skewness and orthogonal quality values in Ansys software can be seen in tables 3 and 4.

Table 3. Skewness mesh metrics spectrum

Excellent	Very good	Good	Acceptable	Bad	Unacceptable
0-0.25	0.25-0.50	0.50-0.80	0.80-0.94	0.95-0.97	0.98-1.00

Table 4. Orthogonal quality mesh metrics spectrum

Unacceptable	Bad	Acceptable	Good	Very good	Excellent
0-0.001	0.001-0.14	0.15-0.20	0.20-0.69	0.70-0.95	0.95-1.00

With mesh specifications as shown in table 5, the meshing quality results are quite good because the maximum skewness and minimum orthogonality values meet the specified meshing standards, namely that the minimum mesh quality is in the acceptable category, so that the simulation can be run to obtain accurate simulation results.

Table 5. Mesh quality on geometry

Geometry	Maximum skewness	Minimum <i>orthogonal</i>	Number of cells
Body domain 2023	0.71	0.16	481426
Body domain 2024	0.51	0.17	372634

3.2 Analysis of drag coefficient values

This study examines aerodynamic improvements based on a comparison of drag coefficient values between the 2023 and 2024 prototype body designs of Mataram University based on varying speeds using the CFD (computational fluid dynamics) method. The drag coefficient (C_d) and drag force (F_D) values are the most influential factors in vehicle design, as they can affect the speed of the vehicle and therefore its efficiency, Rizsal and Martinus (2021). The simulation was carried out until the graph converged, as shown in figure 4, to obtain the drag coefficient and drag force values for each design at varying speeds, as shown in table 6.

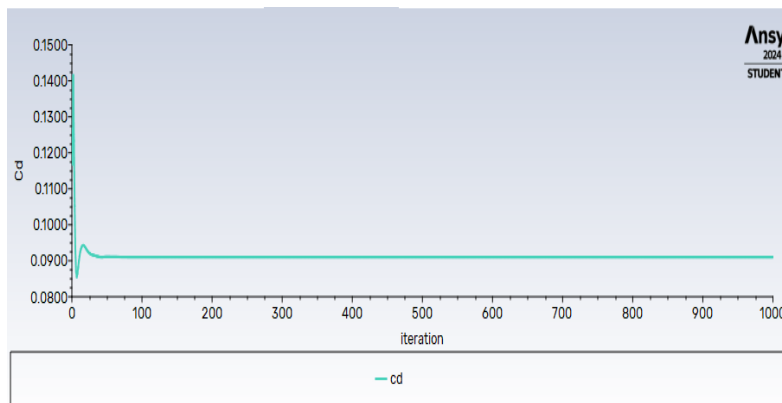


Figure 4. Drag coefficient (C_d) value graph

Table 6. Drag coefficient and drag force values based on speed variations

Body design variations	Speed variation (m/s)	Simulation result parameters	
		C_d	F_D (N)
Body 2023	10.48	0.1071	2.3775
	14.65	0.1007	4.37
	18.82	0.0964	6.9052
	22.98	0.0934	9.9674
	27.15	0.091	13.5616
	Average	0.0977	7.4363
Body 2024	10.48	0.1	1.2922
	14.65	0.0949	2.3957
	18.82	0.0912	3.8024
	22.98	0.0884	5.495
	27.15	0.0862	7.4723
	Average	0.0921	4.0915

Figure 5 shows the relationship between drag force and speed variation for each vehicle body. The lowest drag force value is shown in the 2024 body at a speed of 10.48 m/s, which has a drag force of 1.2922 N. Meanwhile, the highest drag force value is shown in the 2023 body at a speed of 27.15 m/s, which has a drag force of 13.5616 N. Based on the data in figure 5, the drag force value is directly proportional to the vehicle speed. The shape of the vehicle body greatly affects the flow rate around the vehicle body surface, which also affects the aerodynamics of a vehicle, Aprillia and Sriyono (2023). If the shape of the vehicle is not aerodynamic, the drag force and pressure on the vehicle surface will be greater, especially on the front of the vehicle, which will be the first to collide with the fluid. The differences in the drag force values and drag

coefficients are the result of variations in the physical characteristics of each vehicle body, particularly in terms of profile, frontal area, and the front angle of the vehicle, Salman et al. (2025).

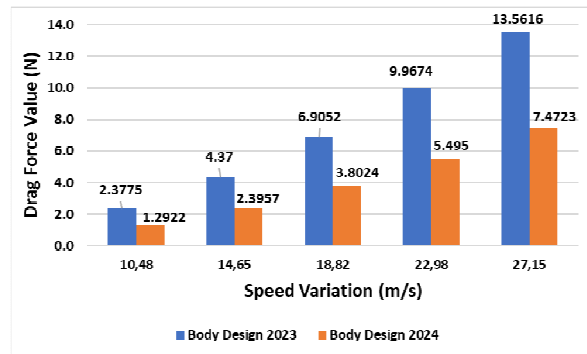


Figure 5. Relationship between drag force and speed variation

Figure 6, shows a graph of the drag coefficient values of both vehicle bodies based on speed variations. It can be seen from the figure that the higher the vehicle speed, the lower the drag coefficient value. A lower drag coefficient value indicates better the aerodynamics of the vehicle because the resistance of the vehicle will be smaller, which will affect the efficiency of the vehicle.

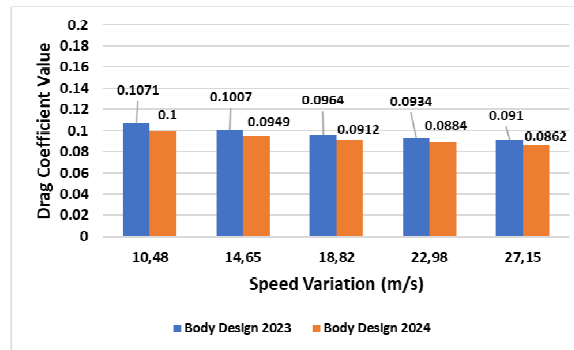


Figure 6. Relationship between drag coefficient and speed variation

3.3 Analysis of vectors, pressure, and velocity at the boundary

After obtaining the drag coefficient value, next step is to analyze the velocity, pressure, and vector contours at the boundary. In this analysis, contours were taken for both body designs at a velocity of 27.15 m/s.

Figure 7, show vector visualizations on the boundary of the design at a speed of 27.15 m/s. The velocity vectors are depicted with arrows indicating magnitude and direction to represent the flow velocity. In addition, velocity vectors are also commonly used to depict areas of irregular turbulence on the surface of a vehicle. The more turbulent areas there are, the greater the drag force will be, Pangestu et al. (2024).

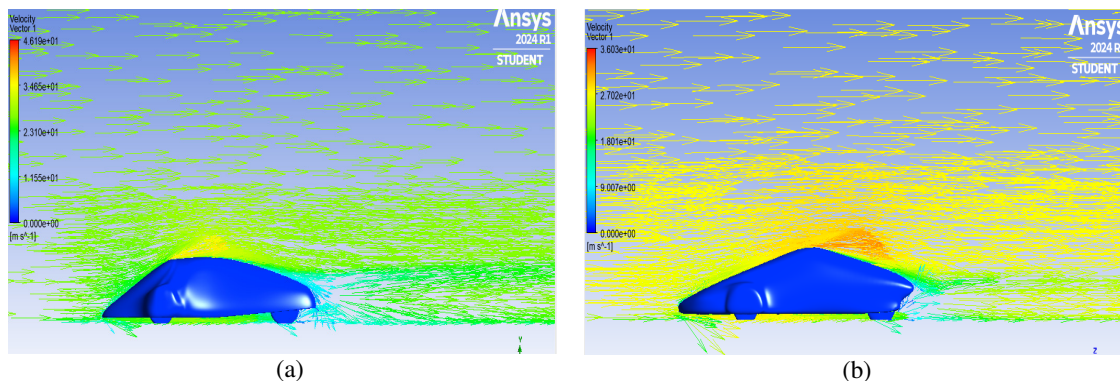


Figure 7. Vector visualization in the simulation: (a) Visualization of the 2023 design vector at a speed of 27.15 m/s, (b) Visualization of the 2024 design vector at a speed of 27.15 m/s

The pressure contours are presented using several color variations. Maximum pressure is marked in red, while low pressure is marked in a darker color, namely blue, as shown in figure 8. It can be seen in the figure that the maximum pressure occurring on the 2024 body is greater, namely 420 Pa, while the maximum pressure for the 2023 body is 409.9 Pa at the same speed of 27.15 m/s. This pressure difference occurs due to the shape of the body and also the surface area of the body in contact with the fluid. From figure 8, the highest average pressure occurs at the front of the vehicle. This occurs because the front of the vehicle is the first part to collide with the fluid, Nurgesang et al. (2021). In addition, the shape of the front of the vehicle and the fluid velocity also affect the amount of pressure exerted on the vehicle.

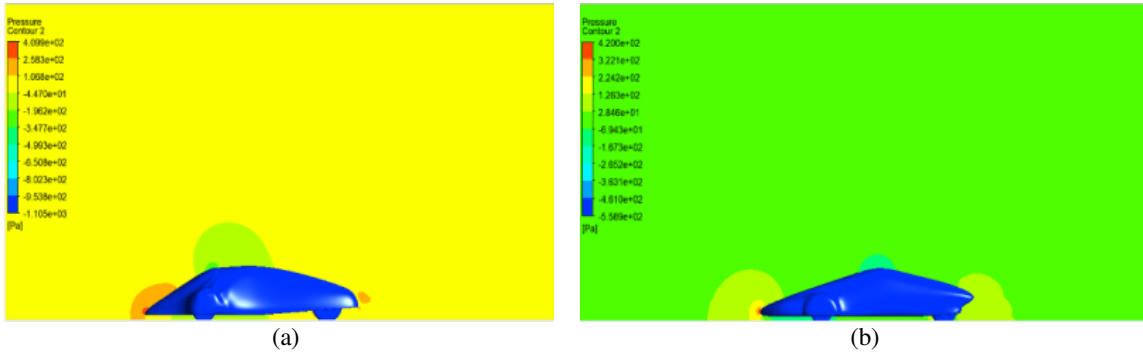


Figure 8. Pressure contour in the simulation: (a) Contour of the 2023 design pressure at a speed of 27.15 m/s, (b) Contour of the 2024 design pressure at a speed of 27.15 m/s

On the velocity contour, the visualization of velocity is almost the same as the pressure contour, which uses a color scale to represent differences in velocity. In figures 9, the highest velocity contour is marked in red, while the lowest velocity is marked in blue. In these figures, it can be seen that the highest velocity is at the top of the prototype. This is due to the shape of the vehicle, which causes an increase in velocity in that area. In addition, there is a decrease in speed at the rear of the vehicle due to irregular airflow. This causes a wake phenomenon at the rear of the vehicle, resulting in increased drag force, Sudaryono (2022).

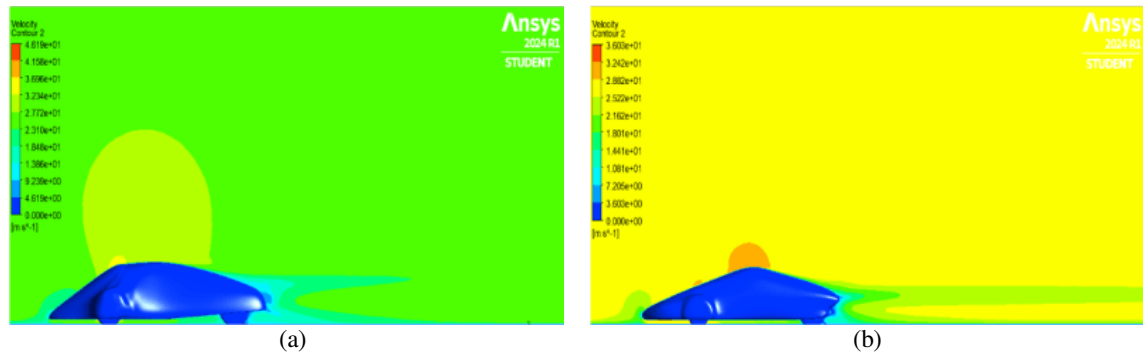


Figure 9. Velocity contour in the simulation:(a) Contour velocity design 2023 at a speed of 27.15 m/s, (b) Contour velocity design 2024 at a speed of 27.15 m/s

4. CONCLUSION

Based on the results of the research conducted, a decrease in the drag coefficient value was obtained in the body design of the 2024 Mataram University prototype vehicle at each varied speed. At a speed of 10.48 m/s, there was a decrease in the drag coefficient of 6.629% from 0.1071 to 0.1. At a speed of 14.65 m/s, there was a decrease of 5.76% from 0.1007 to 0.0949. at a speed of 18.82 m/s, a decrease of 5.394% was obtained from 0.0964 to 0.0912, at a speed of 22.98 m/s, there was a decrease of 5.353% from 0.0934 to 0.0884, and at a speed of 27.15 m/s, there was a decrease of 5.275% from 0.091 to 0.0862. Differences in physical body characteristics across individual vehicles, such as profile, frontal area, and front angle, significantly influence the resulting variations in drag force and drag coefficient values. The average decrease in drag coefficient at each speed variation is 5.682%, so based on these results, it can be concluded that the body design of the 2024 Mataram University prototype vehicle is better than the 2023 body design because the drag coefficient value of the 2024 body design is lower than that of the 2023 design, which means that if the drag coefficient value is low, the

aerodynamics of the vehicle will increase. Therefore, if the aerodynamics are improved, the force that hinders the vehicle will be reduced, resulting in high energy efficiency and more fuel efficiency.

ACKNOWLEDGMENTS

The author would like to thank the Computer Laboratory of the Faculty of Engineering, Mataram University, for enabling the author to conduct research on simulation using the CFD (computational fluid dynamics) method. In addition, the author would like to express their gratitude to their supervisor and the Mandalika Desantara team at Mataram University, as well as all parties involved in the completion of this research.

REFERENCES

- Amaluddin, M.N.H., Analisis koefisien hambatan pada desain bodi kendaraan tipe concept car menggunakan perangkat lunak cfd fluent, *Journal of Energy, Materials, & Manufacturing Technology*, 1(1), 28–33, 2022.
- Ananda, R.P., Akbar, A., Pramesti, Y., Analisis aerodinamika bodi kendaraan KMHE Jayabaya prototype 2.0, *Prosiding SEMNAS INOTEK (Seminar Nasional Inovasi Teknologi)*, 5(3), 218–223, 2021.
- Andriyani, Y.P., Munastha, K.A., Analisis potensi dan pemetaan teknologi turbin angin di seluruh Indonesia, *Rekayasa Industri dan Mesin (ReTIMS)*, 3(2), 77–82, 2022.
- Ansys Inc., Lecture 7: mesh quality & advanced topics, www.featips.com, 2015 (cited: 16 January 2025).
- Aprillia, R.N., Sriyono, R., Analisis aerodinamika bodi kendaraan urban single seater menggunakan metode computational fluid dynamics (CFD), *AEEJ: Journal of Automotive Engineering and Vocational Education*, 4(2), 95–102, 2023.
- Badan Pusat Statistik, Perkembangan jumlah kendaraan bermotor menurut jenis (unit), 2021-2022, www.bps.go.id, 2024 (cited: 28 December 2024).
- Dwi, A., Fawaid, M., Abdillah, H., Desain dan pembuatan bodi mobil Listrik menggunakan fiberglass dengan metode hand lays up, *Jurnal Taman Vokasi*, 9(1), 49–54, 2021.
- Firmanto, B., Merawat Indonesia dari udara: studi numerika erodinamika sayap airfoil NACA, *AMU Press* 1(1), 1–134, 2023.
- Hanafi, A.F., Wardhana, P.B.W., Umar, M.L., Finali, A., Saputra, W., Desain dan analisaero dinamis body mobil hemat energi Jogopati tipe prototype menggunakan metode computational fluid dynamics, *SINERGI POLMED: Jurnal Ilmiah Teknik Mesin*, 5(2), 100–112, 2024.
- Hanif, A. H., Utomo, M. S. K. T. S., Simulasi aerodinamika desain mobil hemat bahan bakar ‘antawirya’ dengan metode computational fluid dynamics, *Jurnal Teknik Mesin*, 3(2), 188–197, 2015.
- Kurnia, A., Efekrumah kaca oleh kendaraan bermotor GRAVITASI: *Jurnal Pendidikan Fisika dan Sains*, 4(2), 1–9, 2021.
- Nurcahyo, Y.E., Wahyudi, P.L., Rancang bangun body fiber carbon dan simulasi aerodinamis dengan ansys untuk mobil hemat energi kategori prototype, *Jurnal Engine: Energi, Manufaktur, dan Material*, 5(2), 90–96, 2021.
- Nurgesang, F.A., Ridlwan, M., Imansyah, Y.T., Desain, analisis aerodinamika, dan pemodelan mobil mikro berdasarkan antropometri tubuh orang Indonesia, *Jurnal Teknik Mesin Indonesia*, 16(1), 35–43, 2021.
- Pangestu, R.A., Asnawi, A., Nayan, A., Setiawan, A., Muhammad, M., Analisa aerodinamika body mobil Listrik dengan metode computational fluid dynamic (cfd) pada variasi frontal area dan kecepatan aliran udara menggunakan software ansys fluent, *Malikussaleh Journal of Mechanical Science and Technology*, 8(1), 175–182, 2024.
- Prastyo, B. W., Syafa’at, I., Dzulfikar, M., Analisis aerodinamika pada bodi mobil hemat energi lintang Samudra menggunakan metode computational fluid dynamics, *Jurnal Ilmiah Momentum*, 16(1), 80–86, 2020.
- Rahmadania, N., Pemanasan global penyebab efek rumah kaca dan penanggulangannya, *Jurnal Ilmu Teknik*, 2(3), 1–13, 2022.
- Riszal, A., Martinus, M., Analisis drag force dan aliran fluida pada desain mobil listrik green campus UNILA berdasarkan kecepatan, *Open Science and Technology*, 1(2), 181–189, 2021.
- Rois, Z.I.H., Dzulfikar, M., Pemodelan kendaraan kontes mobil hemat energi 2021 berdasarkan prediksi konsumsi bahan bakar, *Prosiding Sains Nasional dan Teknologi*, 1(1), 44–48, 2021.
- Salman, S.S., Rachmanto, T., Yudhyadi, I.G.D.K., Analisis aerodinamika dan gaya pada mobil Listrik himpunan Teknik mesin ft Unram dengan metode computational fluid dynamic (CFD), *Prosiding SAINTEK*, 7(1), 298–306, 2025.
- Setyono, A.E., Kiono, B.F.T., Dari energi fosil menuju energi terbarukan: potret kondisi minyak dan gas bumi Indonesia tahun 2020–2050, *Jurnal Energi Baru Dan Terbarukan*, 2(3), 154–162, 2021.
- Sudaryono, A., Analisis aerodinamis bodi mobil listrik Geni Biru KMHE 2020 terhadap kecepatan menggunakan simulasi CFD, *Journal of New Energies and Manufacturing*, 1(1), 41–56, 2022.

- Suryady, S., Zhafran, R., Analisa desain bodi kendaraan tipe urban concept pada pengaruh koefisien drag dan koefisien lift, *Presisi*, 24(1), 74–84,2022.
- Tira, H.S., Muliawan, M.A., Syahrul, S., Numerical analysis of the drag coefficient on energy-efficient vehicle prototypes, *ROTASI*, 25(3), 46–51,2023.
- Wibowo, M.Y., Maulana, I., Ghyferi, A.A., Kurniawan, B.A., Nuril, M., Perancangan chassis prototype mobil Warak dan simulasi statik dengan metode finite element analysis, *Jurnal Mekanik Terapan*, 3(3), 86–92,2022.
- Widiarta, I.P., Suarda, M., Sucipta, M., Sukadana, I.G.K., Simulasi CFD pertukaran udara di ruang Tindakan klinik kesehatan, *Jurnal METTEK*, 8(2), 83–92,2022.