EFFECT OF BODY DESIGN ON AERODYNAMIC PERFORMANCE OF ENERGY-EFFICIENT PROTOTYPE CARS: CFD SIMULATION IN SIMSCALE

[1]Fajar Wendra Krana,[*2]Desmarita Leni,[3]Muchlisinalahuddin
[4]Yuni Vadila

[1,2,3]Mechanical Engineering, Engineering, Universitas Muhammadiyah Sumatera Barat, Indonesia
[4]Mechanical Engineering, Engineering, Universitas Negeri Padang, Indonesia

Author's email:
[1]faejhar.wendra@gmail.com; [2] desmaritaleni@gmail.com; [3]muchlisinalahuddin.umsumbar@gmail.com
[4]yunifadilabkt@gmail.com

**Corresponding author: desmaritaleni@gmail.com*

Abstract. *Energy efficiency in vehicles is a strategic issue in the development of modern transportation technology. Aerodynamic drag is a major factor affecting energy consumption, especially in light prototype vehicles with low to medium speeds. The geometric design of the vehicle body plays a crucial role in reducing the drag coefficient (Cd), which directly impacts energy efficiency and vehicle stability. This study aims to evaluate various body designs for energy-efficient prototype vehicles to minimize drag using a Computational Fluid Dynamics (CFD) simulation method based on the SimScale platform. The simulation results show that a design with a tapered stern, smooth surfaces, and integrated wheel covers can reduce the Cd value by more than 25% compared to conventional designs. In conclusion, optimizing the vehicle body shape by considering aerodynamic principles is effective in improving energy efficiency and is recommended for application in energy-efficient competition vehicles such as the Energy Efficient Car Contest (KMHE) or the Shell Eco-marathon.*

Keywords: *Aerodynamics, CFD, Energy Efficient Cars, Sim Scale, Vehicle Body Design.*

1. INTRODUCTION

Energy efficiency in vehicles is a strategic issue that continues to be a focus in the development of modern transportation technology. Aerodynamic drag is a major component of energy consumption, particularly in low- to medium-speed prototype vehicles with lightweight weights.(Rauf et al., 2022)The geometric design of the vehicle body directly affects the size of the drag force that occurs when the vehicle moves through the air flow.(Hanafi et al., 2024).

The application of aerodynamic principles to vehicle design plays a significant role in reducing the drag coefficient (Cd), which directly impacts fuel efficiency and reduces the power consumption of electric motors in electric vehicles. High air resistance causes vehicles to require more energy to maintain speed, especially at medium to high speeds.(Salman et al., 2024)Therefore, Cd reduction is a key strategy in energy-efficient vehicle design. Optimizing the vehicle body shape is the most effective engineering solution, focusing on key elements such as windshield angle, roof curvature, rear tapering, and wheel cover integration that can smooth airflow around the wheels.(Padang State Polytechnic et al., 2023).

Experimental studies and design simulations show that simple geometric changes to the area can reduce drag by more than 30% compared to conventional square or boxy designs.(Salman et al., 2024)This effect not only reduces engine workload but also increases vehicle range, particularly for electric vehicles and prototypes with limited energy capacity. Furthermore, aerodynamic design also impacts vehicle stability against side winds and turbulence, thus contributing to safety and driving comfort.(Salman et al., 2024). Thus, vehicle body design can no longer be viewed solely as an aesthetic element or mechanical protection, but rather as a key functional variable that determines the vehicle's overall energy performance. Integration of aerodynamic, aesthetic, and structural functionality considerations is essential in the development of modern energy-efficient vehicles.

The use of Computational Fluid Dynamics (CFD) has become an effective approach for evaluating airflow characteristics prior to physical prototyping. This method is capable of representing flow phenomena, pressure distribution, and wake regions in detail on vehicle surfaces.(Salman et al., 2024). The SimScale platform, as a cloud-based CFD simulation tool, provides easy access for researchers and students to conduct vehicle aerodynamic analysis without the need for complex local computing infrastructure (SimScale, 2024).

The urgency of this study focuses on improving the aerodynamic efficiency of an energy-efficient prototype car through optimal body design. The need for a design that can minimize air resistance is crucial, especially for the development of competition vehicles such as those in the Energy Efficient Car Contest (KMHE) or the Shell Eco-marathon. Without mastery of aerodynamic principles and design validation based on simulation data, designing high-efficiency vehicles will struggle to achieve optimal performance targets.(Rizaly & Romadhoni, 2024).

This research is expected to contribute to the selection of optimal body designs that support overall vehicle performance, particularly in the context of energy saving and engineering design based on the latest numerical data.

2. LITERATURE REVIEW

2.1 Energy Efficient Cars

Energy-efficient cars are designed to optimize fuel use by reducing aerodynamic drag, minimizing vehicle weight, and increasing drivetrain efficiency.(D Ieni, 2023)In vehicle aerodynamic studies, the drag coefficient (Cd) is an important parameter that reflects the amount of air resistance a vehicle experiences while moving. Aerodynamic designs, such as rounded body shapes, optimally angled windshields, and low ground clearance, contribute significantly to fuel efficiency. The use of technologies such as spoilers and diffusers also plays a role in directing airflow to reduce turbulence and create downforce.(Department of Mechanical Engineering, Universitas AKPRIND Indonesia, Indonesia et al., 2024).

2.2 Computational Fluid Dynamics (CFD)

CFD is a numerical simulation method used to analyse the behaviour of fluid flows, including air, around vehicles. The basic principles of CFD involve the conservation of mass, momentum, and energy, expressed through the Navier-Stokes equations. CFD allows the simulation of laminar and turbulent flows by utilizing turbulence models such as k-ε, k-ω, and SST. In vehicle design, CFD is used to predict pressure distributions, flow patterns, and drag and lift forces without the need for expensive physical testing.(Gde Didit Citra Anggarana & Made Gatot Karohika, 2022).

2.3 Sim Scale as a CFD Simulation Tool

Sim Scale is a cloud-based platform that enables engineering analysis without the need for high-spec hardware. Sim Scale supports fluid flow simulations using advanced solvers like Open FOAM and provides interactive visualization of the results. Features such as automated meshing, velocity contours, and aerodynamic parameter calculations make it highly relevant for analysing the performance of energy-efficient vehicles. In this study, Sim Scale was used to evaluate a prototype design for a diesel-engined vehicle by varying the airflow velocity and comparing several geometric designs.(Haryadi et al., 2023).

2.4 Vehicle Body Design and Aerodynamic Elements

Modern vehicle body designs integrate aerodynamic principles to improve fuel efficiency, stability, and ride comfort. Key elements contributing to aerodynamic performance include:

1. Spoiler and Fins

The main element is the spoiler, which reduces lift and increases vehicle stability by creating downforce. Additionally, fins such as canards and side skirts help direct airflow to minimize turbulence, thus supporting vehicle stability at high speeds.(Luthfie et al., 2019).

2. Shape and Contour

The vehicle's shape and contours are also highly considered. A rounded body design with smooth transitions between the front, roof, and rear helps reduce airflow separation and wake turbulence. This not only improves aerodynamic efficiency but also reduces drag.(Mail, 2020).

3. Ground Clearance and Underbody Design

Other influencing factors include ground clearance and undercarriage design. Vehicles with low ground clearance, combined with flat undercarriages, significantly reduce drag and lift. The addition of a diffuser at the rear of the vehicle also helps create a more stable airflow beneath the vehicle, producing downforce that supports stability at high speeds.(Mail, 2020).

4. Windshield and Roof Slope

The slope of the windshield and roof also affects the vehicle's aerodynamic performance. Optimal slope angles ensure airflow adheres to the vehicle's surfaces, reducing the risk of separation and turbulence, and improving the overall efficiency of the vehicle's aerodynamic design. Integrating these elements is essential for developing fuel-efficient and stable vehicles under various driving conditions.(Ikhsan Alfajri et al., 2020).

2.5 Research Relevance

This research focuses on evaluating the aerodynamic performance of energy-efficient vehicles using CFD simulations. By integrating aerodynamic design elements and utilizing the SimScale platform, this study aims to identify the optimal vehicle body design in terms of fuel efficiency and stability. Airflow analysis at varying speeds and drag and lift calculations are crucial steps in this process.(Nurcahyo & Wahyudi, 2021).

3. RESEARCH METHODS

This study uses a quantitative experimental method based on Computational Fluid Dynamics (CFD) simulations to analyze the performance of an energy-efficient prototype vehicle body design. This method was chosen because it allows for controlled and detailed observation of simulation results on parameters such as drag coefficient (Cd), flow velocity, and pressure distribution, without the need for a physical prototype or wind tunnel testing. The study was conducted online through the cloud-based SimScale platform, thus not being tied to a specific physical location. The study took place from January to July 2025, encompassing literature review, model design, simulation, data analysis, and report preparation.

This research goes through several stages as shown in the research flow chart below:

Figure 1. Flow chart

3.1 Literature Study

The initial stage of this research began with a literature review to understand relevant basic theories. Researchers studied the concept of vehicle aerodynamics, the basic principles of Computational Fluid Dynamics (CFD), and the workings of the cloud-based simulation platform Sim Scale. Furthermore, they reviewed various previous studies addressing energy-efficient cars and CFD simulation approaches to aerodynamic design to identify research gaps and strengthen the theoretical foundation.

3.2 Design

The design stage is the process of designing the vehicle body shape by considering aerodynamic parameters to reduce air resistance, thereby supporting energy efficiency in energy-efficient vehicles.

3.3 CFD Simulation

In the next stage, the simulation of the vehicle body design was imported into Sim Scale geometry. Then, the meshing process was performed, which involves creating a numerical grid as the basis for solving the CFD equations. Researchers ensured the mesh quality was good to avoid errors in the numerical calculations.

3.4 Simulation Results Analysis and discussion

All simulation results were collected and compared in graphs and tables. The primary focus of the analysis was comparing the drag coefficient values of each design. Lower drag values indicate a more aerodynamic and energy-efficient design.

a. Best Design Evaluation

Based on the comparison results, the body design with the lowest air resistance will be identified as the best. Researchers will then evaluate how this design improves aerodynamic efficiency and energy savings in the prototype vehicle.

b. Discussion of Results

The simulation and analysis results are then comprehensively discussed. Researchers compare the findings with previous research to determine their suitability or discrepancies. This discussion includes design implications for the practical development of energy-efficient vehicles.

c. Validation of Results

Validation is carried out as the final stage in the analysis process to ensure that the results obtained are not only numerically valid, but also correspond to theory and empirical evidence.

3.5 Conclusion

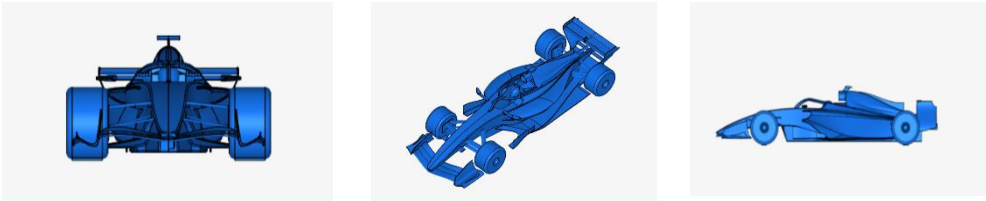
In the conclusion stage, the researcher summarizes the results of the entire research process, starting from problem identification and theoretical analysis to the design of an energy-efficient vehicle body. The conclusion emphasizes the achievement of the research objective, which is to produce a body design that theoretically minimizes aerodynamic drag. Furthermore, this stage also presents the design's implications for vehicle energy efficiency and provides recommendations for further research to validate the design's performance.

4. RESULTS AND DISCUSSION

4.1 Results

This research focuses on the design and analysis of an energy-efficient prototype vehicle body to improve aerodynamic efficiency. The design was conducted through three-dimensional (3D) modeling using CAD software, considering aerodynamic aspects and geometric simplicity to support the simulation process.

The prototype car design used in this study is shown in **Figure 2.** The model was developed to minimize air resistance by smoothing the body contours and paying attention to the air flow around the vehicle.



(a) (b) (c)

Figure 2. Prototype Car Design (a) Front View, (b) Isometric View, and (c) Side View

In addition, the flow region is also comprehensively modeled, as seen in **Figure 3.** This flow region is useful for representing the airflow conditions around the vehicle during the simulation. This aims to obtain realistic pressure and velocity distributions in aerodynamic performance analysis.

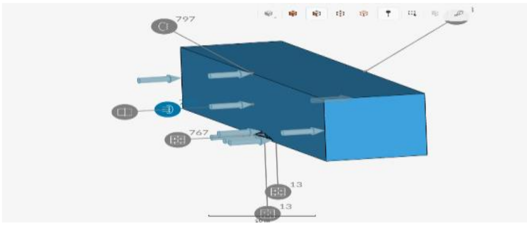


Figure 3. Flow Reaction

Next, a simulation was performed on the vehicle body design that had been created and imported into SimScale geometry. The meshing process, which involves creating a numerical grid as the basis for solving the CFD equations, was performed. Researchers ensured the mesh quality was good to avoid errors in the numerical calculations.

Mesh creation occurs after the domain creation process is complete. The meshing process involves various steps and parameter settings. The area or volume of fluid that has been divided into small cells is referred to as a mesh. The mesh division in this model can be seen in Figure 4.

Figure 4. *Meshing* Simulation

The mesh division of the prototype car model used in the simulation process is shown in Table 1. This division is designed to achieve a balance between the accuracy of the simulation results and the efficiency of computing time.

Table 1. Car Design Meshing Division

Criteria	Mark
This mesh meets the quality criteria.	
Number of nodes	7,768,141
Volume amount	25,826,125
Number of quadrangles	11,899,092
Number of faces	57,070,829
Number of triangles	45,171,737
Number of hexahedrons	1,133,592
Number of edges	74,742
Number of tetrahedrons	17,930,139
Process completed	
Number of pyramids	1,650,121
Number of prisms	5,112,273

In this study, a fluid domain was created as a boundary condition to analyze fluid flow through a vehicle body in a wind tunnel. The boundary conditions in this model include the vehicle body, the walls at the top, bottom, right, and left, as well as the inlet for incoming air flow and the outlet as the analysis boundaries at the top and bottom walls. The division of boundary conditions in this model can be seen in Table 2.

Table 2. Boundary Condition Table

No.	Name	Type***Boundary***
1	*Inlet*	*Inlet velocity*
2	*Outlet*	*Pressure outlet*
3	*Body*	*Wall*
4	*Top*	*Wall*
5	*Bottom*	*Wall*
6	*Right*	*Wall*
7	*Left*	*Wall*

The division of boundary conditions in the simulation domain is shown in Figure 5. In this model, the inlet condition is set at the front side of the domain to simulate the inlet flow velocity, while the rear side is given an outlet condition to allow the flow to exit. The vehicle surface and the bottom surface of the domain are given a no-slip wall condition, while the top and sides are given a symmetry condition to reduce the influence of the domain boundary on the simulation results. The determination of these boundary conditions is done to ensure an accurate physical representation of the flow around the prototype car.

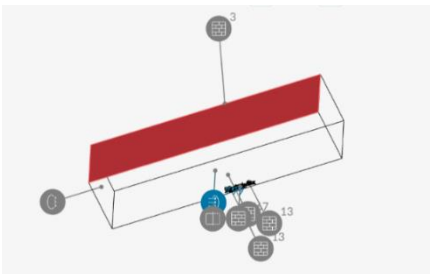


Figure 5. Design Simulation Boundary Conditions

The simulations were conducted using the SimScale platform using an incompressible flow and steady-state approach. Key parameters such as flow velocity, fluid type, boundary conditions, and turbulence model were systematically determined to ensure the validity and reliability of the simulation results. The setup parameters used in the CFD simulation of the design are listed in Table 3.

Table 3. Simulation Parameters

Category	Parameter	Values / Settings
Physics Model	Flow	Incompressible
	Turbulence model	*k-omega SST*
	Time dependency	Steady-state
	Algorithm	Simple
	Passive species	0 (zero)
Fluid (Material)	Type of fluid	Air
	Density (ρ)	1.225 kg/m³ (default)
	Dynamic viscosity (μ)	1.81 × 10^{-5}Pa·s (default)
Flow Rate	Inlet velocity (U)	100 m/s
Pressure	Outlet pressure	0 Pa (gauge pressure)
Boundary Conditions	Inlet	Inlet velocity = 100 m/s
	Outlet	Outlet pressure = 0 Pa
	Vehicle wall	*Wall – No-slip*
	Lateral & upper/lower domain walls	*Wall – Slip*
Flow Domain	Domain area (p × l = 400.2 m²)	20 m (w) × 10 m (w) x 20 m = 400 m²
Mesh	Mesh refinement	Fine near vehicle surface (level 5–7), coarser elsewhere
	Mesh type	Automatic Hex-dominant or Tet-dominant
Simulation	Type	Steady-state CFD
	Solver	Simple

Based on the established parameters, a CFD simulation was run on the prototype energy-efficient car body design. The simulation results were then analyzed and explained in the following section.

1. Velocity Magnitude

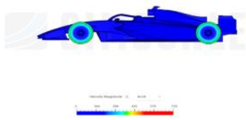


Figure 6. Velocity Magnitude

The velocity magnitudes in Figure 6 show the distribution of velocity magnitudes around the prototype energy-efficient car. The highest airflow velocity of 720 km/h is observed around the wheels, indicating significant aerodynamic drag. Meanwhile, a steady flow with a low velocity of 0–144 km/h dominates the main body of the vehicle, reflecting the design's efficiency in directing airflow.

2. Vorticity Magnitude

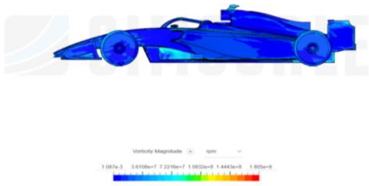


Figure 7. Vorticity Magnitude

The CFD simulation results in Figure 7 show that the body design of the energy-efficient car prototype significantly affects the distribution of vorticity magnitude. The dark blue area with a low vorticity of 1.087 × 10^{-3} rpm indicates a stable airflow, while the yellow to red area with a high vorticity of 1.805 × 10^{8} rpm is found around the wheels and rear of the vehicle, indicating turbulence and aerodynamic drag.

3. Wall Shear Stress Magnitude

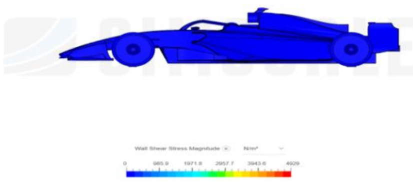


Figure 8. Wall Shear Stress Magnitude

In the simulation results of Figure 8, the distribution of Wall Shear Stress Magnitude on the vehicle surface in this simulation shows the variation of wall shear stress values in units of N/m2, as represented by the color scale. The shear stress values range from 985.9 N/m2 - 6,020 N/m2. Areas with low shear stress values of 985.9 N/m2, marked in dark blue, dominate the vehicle surface that is not exposed to direct air flow, such as the upper part of the body. Medium stress values of 2,057.7 N/m2 - 3,846.2 N/m2, marked in green-yellow occur in transition areas, such as around the wheels and curved surfaces. Meanwhile, the maximum value is at 6,020 N/m2, marked in red) detected in zones with fast air flow and high turbulence, such as at the sharp edges and rear areas of the vehicle.

4. Pressure

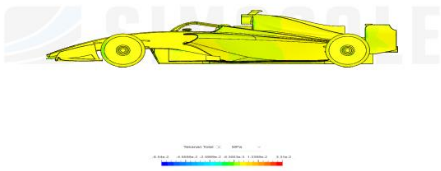


Figure 9. Pressure

The simulation in Figure 9 shows the Pressure distribution on the vehicle surface, with a range of values between (blue) to (red). Low pressure of −0.0654 MPa is observed in certain areas, especially at the rear of the vehicle, due to the wake phenomenon and low dynamic pressure. Medium pressure of −0.0259 MPa to 0.0134 MPa dominates the vehicle surface exposed to laminar airflow. High pressure of -0.0331 MPa is concentrated in the front area of the vehicle, such as the nose and frontal surface, where airflow stagnation occurs.

5. Specific Dissipation Rate(ω)

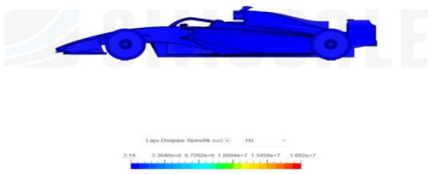


Figure 10. Specific Dissipation Rate(ω)

The CFD simulation results in Figure 10 show the distribution of the specific dissipation rate (ω) around the body of the energy-efficient prototype vehicle. The highest specific dissipation rate values (up to 1.68×10^7 Hz) were detected in the rear wheel and diffuser areas, indicating the presence of strong vortices and potential flow separation that increase aerodynamic drag. Meanwhile, very low ω values (3.14 Hz) dominate the front section of the vehicle, reflecting relatively stable flow and an aerodynamically efficient design in the nose and front wing regions. Moderate turbulence distribution (3.36×10^6–1.00×10^7 Hz) was observed along the sides of the vehicle body, particularly in the cockpit and side pod areas, contributing to increased surface friction.*4.2 Discussion*

In energy-efficient vehicle research, aerodynamic efficiency is a vital component because it directly affects the amount of energy required to maintain vehicle speed, especially at low to medium speeds, where air resistance remains significant compared to other forces.

Using a CFD simulation approach, various physical parameters of fluid flow around the vehicle are analyzed. The results of these simulations are then interpreted to identify the extent to which body design contributes to improving or reducing aerodynamic efficiency.

1. The Effect of Body Design on Flow Characteristics and Drag Force

The design of the vehicle body directly affects the distribution of fluid flow around the vehicle's surface, and in the context of energy-efficient vehicles, this design plays a crucial role in suppressing drag, which is a major contributor to energy consumption. Based on the results of the CFD simulations, it was found that the streamlined geometry of the vehicle's main body is able to maintain airflow following the surface smoothly (attached flow), with a flow speed range between 0 and 144 km/h. This indicates that the design with rounded contours and smooth surface transitions is effective in maintaining laminar flow and avoiding flow separation, which is generally the main cause of high drag.

However, the unintegrated design, particularly around the front and rear wheels, resulted in extreme flow accelerations of up to 720 km/h. This phenomenon indicates a constriction in the flow path, which increases the velocity gradient and creates a localized turbulence zone. The high speeds around the wheels also trigger boundary layer disturbances, leading to flow separation and the formation of intense vortices.

The simulations also showed very high vorticity of up to 1.805×10^8 rpm in the rear area of the vehicle and around the wheels, indicating the formation of a large and uncontrolled wake region. This wake is the main cause of pressure drag, which is a drag force that originates from the pressure difference between the front and rear of the vehicle. The pressure simulation data showed high pressure on the frontal surface (+0.0331 MPa) due to flow stagnation, and low pressure (−0.0654 MPa) at the rear of the vehicle as a result of flow detachment. This pressure difference creates significant form drag, contributing directly to the increase in total vehicle drag.

In addition, the wall shear stress distribution reached a maximum value of 6,020 N/m², especially at sharp edges and the rear area of the vehicle. This value indicates that the body design with its uneven contours creates high skin friction drag due to the frictional force between the fluid and the solid surface. This is exacerbated by the high specific dissipation rate (ω) of up to 1.68×10^7 Hz, indicating the level of kinetic energy lost in the form of turbulence in the wake region. This energy is a direct loss of the vehicle's potential motion, thus worsening energy efficiency.

Overall, these results reinforce the finding that un-aerodynamic body designs, such as exposed wheel arches and a non-smoothly tapered rear end, are major contributing factors to increased total drag. In a prototype vehicle that relies on maximum energy efficiency, this is particularly detrimental, as any increase in drag immediately increases energy consumption. Therefore, body design should prioritize smooth contour integration, enclosing the wheel arches, and gradually tapering the rear end to minimize wake and pressure differences, as well as energy losses due to shear stress and turbulence.

CFD simulation results indicate that body design optimization plays a strategic role in improving the aerodynamic efficiency of energy-efficient prototype vehicles. One significant improvement recommendation is the addition of aerodynamic wheel covers, such as wheel covers or fairings, which have been shown to reduce airflow acceleration and vortex formation around the wheels. This effort can reduce drag originating from flow instability in that area. In addition, the gradually tapered rear of the vehicle design can reduce negative pressure in the wake region, thereby reducing the overall pressure drag component. On the other hand, the smooth body surface contours and gentle corner transitions contribute to suppressing wall shear stress, while minimizing energy loss due to turbulence. Overall, this design approach supports the achievement of more efficient aerodynamic performance and is in line with the basic principles of energy-efficient vehicle development.

3. Validation of Research Results

The results of Computational Fluid Dynamics (CFD)-based numerical simulations in this study indicate that optimizing the body design of an energy-efficient prototype vehicle has a significant impact on reducing drag and increasing aerodynamic efficiency. To verify these findings, external validation was conducted by comparing the simulation results with several recent studies over the past 10 years that examined vehicle aerodynamic characteristics using a similar approach. One key aspect is the use of wheel covers, which in a study by Putra et al. (2021) were shown to reduce the drag coefficient by up to 12% by stabilizing airflow around the wheels, reducing vortices, and reducing pressure drag. Similar findings were also reported by Al-Mashhadany et al. (2020) who stated that wheel fairings produce more homogeneous airflow and reduce the wake area. In addition, modifying the rear geometry of the vehicle with a tapered contour or the Kammback model has been shown to be effective in suppressing turbulence formation in the rear area of the vehicle, as confirmed by a study by Aulia et al. (2017) who recorded a 10.7% reduction in the drag coefficient after implementing a gradual tapered design. A recent study by Wang & Li (2023) showed that smoother vehicle body surfaces significantly slow the transition from laminar to turbulent flow, reduce wall shear stress, and improve flow efficiency at high speeds. Comparing the results of this study with the literature, it can be concluded that the implemented design approach has been scientifically tested and is valid in improving the aerodynamic performance of energy-efficient vehicles.

CONCLUSION

This study demonstrates that the body design of an energy-efficient prototype vehicle significantly impacts airflow characteristics and aerodynamic efficiency. Computational Fluid Dynamics (CFD) simulation results show that the highest airflow velocity occurs around the wheels, indicating significant aerodynamic drag in that area. On the other hand, stable flow and low velocity dominate the main body of the vehicle, reflecting the success of the streamlined design in directing the flow. Furthermore, areas with high vorticity and shear stress values are identified at the rear and around the wheels, indicating the center of turbulence and drag formation. The pressure distribution indicates stagnation pressure at the front and negative pressure at the rear of the vehicle, which contribute to the total drag force. The high turbulent energy dissipation rate in the diffuser area also reinforces the flow separation at the rear of the vehicle. Overall, the body design used in this study is proven to be able to direct airflow more efficiently and reduce aerodynamic drag, thus supporting the improvement of the energy efficiency of the prototype vehicle.

REFERENCES

D Ieni, HY (2023). Modeling of Car Tire Damage Inspection Using Convolutional Neural Network (CNN). Journal of Materials, Manufacturing and Energy Engineering, 6(2). https://doi.org/10.30596/rmme.v6i2.16198

Department of Mechanical Engineering, Universitas AKPRIND Indonesia, Indonesia, Waluyo, J., Hariyanto, S.D., Department of Mechanical Engineering, Universitas AKPRIND Indonesia, Indonesia, Lestari, N., & Department of Mechanical Engineering, Universitas AKPRIND Indonesia, Indonesia. (2024). Evaluation of Spoiler Design on the Aerodynamic Performance of an Energy-Efficient Car Prototype Body. Engineering and Technology Journal, 09(08). https://doi.org/10.47191/etj/v9i08.49

Gde Didit Citra Anggarana, B., & Made Gatot Karohika, I. (2022). Analysis Of Car Body Aerodynamics With Speed Variations Using Cfd Software. Sibatik Journal: Scientific Journal of Social, Economic, Cultural, Technology, and Education, 1(8), 1455–1462. https://doi.org/10.54443/sibatik.v1i8.192

Hanafi, AF, Wardhana, PBW, Umar, ML, Finali, A., & Saputra, W. (2024). Aerodynamic Design And Analysis Of The Energy-Effective Jogopati Prototype Car Body Using The Computational Fluid Dynamics Method. Sinergi Polmed: Jurnal Ilmiah Teknik Mesin, 5(2), 100–112. https://doi.org/10.51510/sinergipolmed.v5i2.1670

Haryadi, DRB, Sofyan, E., & Eko Prasetiyo, E. (2023). Aerodynamic Analysis of Folding Fin Aerial Rocket 70 mm on Incompressible Flow Using CFD-Based SimScale. Teknika STTKD: Jurnal Teknik, Elektronik, Engine, 9(1), 82–93. https://doi.org/10.56521/teknika.v9i1.844

Ikhsan Alfajri, N., Tjahjana, Ph.D, DDP, & Kristiawan, B. (2020). The Effect of Adding Gurney Flap to Mshd Type Airfoil on Fastback Car with Variations in Gurney Flap Height and Airfoil Angle Tilt Using CFD Modeling Method. Mekanika: Jurnal Ilmiah Mekanika, 19(1), 35. https://doi.org/10.20961/mekanika.v19i1.40005

Luthfie, AA, Romahadi, D., Ghufron, H., & Murtyas, SD (2019). Numerical Simulation On Rear Spoiler Angle Of Mini Mpv Car For Conducting Stability And Safety. Synergy, 24(1), 23. https://doi.org/10.22441/sinergi.2020.1.004

Mail, A. (2020). Ergonomic Study Based On Biomechanical And Physiological Aspects In The Design Of Motorized Tricycles (Bentor) In Makassar City. Journal of Industrial Engineering Management, 5(1), 25–35. https://doi.org/10.33536/jiem.v5i1.549

Nurcahyo, YE, & Wahyudi, PL (2021). Fibercarbon Body Design and Aerodynamic Simulation with Ansys for an Energy-Efficient Prototype Car. Jurnal Engine: Energi, Manufaktur, Dan Materi, 5(2), 90. https://doi.org/10.30588/jeemm.v5i2.883

Padang State Polytechnic, Adriansyah, A., Leni, D., & Sumiati, R. (2023). Comparative analysis of energy-efficient air conditioners based on brand. Polimesin Journal, 21(04). https://doi.org/10.30811/jpl.v21i4.3625

Rauf, W., Rifal, M., & Boli, RH (2022). Computational And Experimental Study Of The Effect Of Active Control On Aerodynamic Drag Of Vehicle Models. Radial: Journal of Science, Engineering, and Technology Civilization, 10(1), 83–94. https://doi.org/10.37971/radial.v10i1.268

Rizaly, A., & Romadhoni, FR (2024). Numerical Simulation and Design Modification According to Shell Eco Marathon Regulations Aerodynamic Performance of Urban Concept Cars. Engineering Energy Systems and Manufacturing (ReSEM), 2(2), 93–100. https://doi.org/10.30651/resem.v2i2.21893

Salman, SS, Rachmanto, T., & Yudhyadi, I. (2024). Aerodynamic And Force Analysis Of An Electric Car From The Mechanical Engineering Society Of Ft Unram Using The Computational Fluid Dynamic (Cfd) Method. Proceedings Of Saintek, 7, 398–306. https://doi.org/10.29303/saintek.v7i1.3425