Automotive Experiences
Vol. 8 No. 2 (2025) pp. 452-463
p-ISSN: 2615-6202 e-ISSN: 2615-6636

Research Paper

Aerodynamic Approach to Two-Passenger City Car Design: A Study of Square Back and Compact Shapes

Randi Purnama Putra[1,2], **Dori Yuvenda**[1,2], **Remon Lapisa**[1,2], **Wanda Afnison**[1,2], **Milana**[1,3] **M. Yasep Setiawan**[1,3], **Ahmad Arif**[1,3], **Dani Harmanto**[4]

[1]Transportation and Vehicle Research Group, Universitas Negeri Padang, Padang 25131, Indonesia
2Department of Mechanical Engineering, Universitas Negeri Padang, Padang 25131, Indonesia
[3]Department of Automotive Engineering, Universitas Negeri Padang, Padang 25131, Indonesia
[4]Department of Aeronautical Engineering, De Monfort University, Leicester LE1 9BH, United Kingdom

randipurnama@ft.unp.ac.id

https://doi.org/10.31603/ae.13686

Published by Automotive Laboratory of Universitas Muhammadiyah Magelang

Article Info
Submitted:
30/05/2025
Revised:
23/09/2025
Accepted:
24/09/2025
Online first:
25/09/2025

Abstract

The development of lightweight electric cars for urban mobility needs requires efficient aerodynamic design without sacrificing space efficiency. This study presents a novel method by investigating the combination of a two-seater city car's compact dimensions and square back shape, which has not been extensively researched about low- to medium-velocity vehicles. This study's objective is to assess the design's aerodynamic performance using numerical simulations using the Computational Fluid Dynamics (CFD) approach. The vehicle model is designed with a compact body and square back, which is commonly used in small vehicles with high maneuverability requirements. The simulations are conducted at three different air velocity levels: 10, 20, and 30 m/s. The results of the study showed an increase in the value of the drag coefficient (*Cd*) along with an increase in flow velocity. At a velocity of 10 m/s, the *Cd* value was recorded at 0.4536. When the velocity increased to 20 m/s, the drag coefficient increased slightly to 0.4563. Further increases in velocity to 30 m/s resulted in a *Cd* value of 0.4581. This *Cd* value shows the consistency of aerodynamic performance with increasing velocity, with fluctuations that remain within the efficiency limits of lightweight vehicles. The pressure distribution contour shows high-pressure accumulation at the front and low pressure at the rear of the vehicle, which generates large turbulent wakes in the rear area and contributes to increased drag. These findings indicate that the square rear body design faces significant aerodynamic challenges. Therefore, design strategies such as adding a rear spoiler, using a rear diffuser, and optimizing the rear body angle are suggested as potential solutions to improve flow efficiency.

Keywords: Aerodynamics study; City car; Square back; Compact design; CFD; Drag coefficient

1. Introduction

In recent years, the advancement of electric vehicles (EVs) has accelerated, coinciding with heightened awareness regarding the necessity of diminishing carbon emissions, enhancing energy efficiency, and reducing reliance on fossil fuels [1], [2], [3]. Large cities around the world face serious challenges related to air pollution, traffic congestion, and limited parking spaces. Therefore, there is a need for vehicles that are not only environmentally friendly but also compact, space-efficient, and adaptable to the characteristics of dense urban traffic [4]. Light electric vehicles, especially those with a capacity of two passengers and designed for low to medium velocity, are one of the ideal solutions for mobility in urban environments [5]. However, the main challenge in designing light electric vehicles is not only related to energy efficiency from the side of the drive system or battery, but also from the aspect of the design of the vehicle body itself, especially aerodynamic performance. Aerodynamic drag contributes significantly to vehicle energy consumption, especially at velocities above 10 m/s [6], [7], [8]. Even at velocities that are relatively low in the context of urban traffic, air drag still affects the overall efficiency of the vehicle. Therefore, efficient aerodynamic design is crucial to increase mileage, reduce battery power consumption, and maintain vehicle stability [9], [10]. The challenge is further complicated when it must be balanced with the need for a compact and functional vehicle shape, especially in a two-passenger configuration.

To address these challenges, much previous research has been conducted to design a more aerodynamic vehicle shape [11], [12], [13], [14]. Various body design approaches have been developed and tested to minimize drag, ranging from streamlined shapes that follow the ideal contour of the airflow, to modifications of body elements such as spoilers, diffusers, and roof shapes [15], [16], [17]. Vehicle body shapes such as fastbacks, notchbacks, and hatchbacks have been the main focus of research because they have relatively good aerodynamic characteristics [18], [19], [20]. For example, the fastback design that tapers at the rear is often associated with a low drag coefficient (C_d) because its shape helps smooth airflow and reduce turbulence.

However, the majority of these studies have focused on medium to large vehicles, or on vehicles targeting high performance, such as modern sports cars and sedans. Meanwhile, studies on the aerodynamics of compact two-passenger vehicles for low to medium velocity are still very limited. In the context of lightweight electric vehicles optimized for urban environments, there is a tendency for space efficiency to be the main priority, and aerodynamic aspects are often compromised. One of the body shapes often used in small vehicles is the square back shape, which has a perpendicular rear and provides maximum space efficiency. This shape is commonly found in microcars, small commercial vehicles, or mini-sized experimental vehicles. However, there are not many studies that explicitly evaluate the aerodynamic performance of the square back shape at low to medium velocities.

This study seeks to address the existing research gap. The square-back configuration is frequently deemed to exhibit suboptimal aerodynamic performance due to the generation of a low-pressure zone at the vehicle's rear, which can augment drag. However, this assumption has not been tested in depth in the context of light electric vehicles operating at low velocity—namely in the range of 10 to 30 m/s. At this velocity, airflow patterns, vortex formation, and pressure distribution can exhibit different behaviors compared to high velocity. In addition, there is the potential that with its very compact dimensions and simple body shape, the square back can provide a reasonable compromise between space efficiency and aerodynamics, especially when applied to vehicles with high maneuverability requirements and limited cabin space.

Therefore, this work presents a strategy to address these problems by assessing the body design of a tiny two-seater car featuring a square back form through numerical simulations utilizing the Computational Fluid Dynamics (CFD) method. This method was chosen because it has a high capability in modeling the complex interactions between airflow and object surfaces in detail, and allows for a variety of velocity test scenarios without requiring a physical prototype. The aerodynamic performance of city vehicles was examined over the operational velocity range by conducting simulations at three different air velocity levels: 10 m/s, 20 m/s, and 30 m/s. The analyzed vehicle design has compact dimensions and simple geometry, which reflects the real needs of efficient future urban vehicles.

2. Methods

2.1. Simulation Method

This study employs a numerical method utilizing CFD Ansys R1 2024 student version to analyze the aerodynamic performance of a two-passenger design characterized by a square back body shape and compact dimensions. The examined vehicle design includes principal dimensions of 2.59 meters in length, 1.6 meters in width, and 1.52 meters in height. These dimensions are designed to accommodate mobility needs in dense urban areas while ensuring maneuvering and parking efficiency. These compact proportions are chosen based on modern urban vehicle design trends that emphasize space efficiency and reducing air resistance without sacrificing user comfort (Figure 1).

In this simulation, the fluid used is air as the external flow medium, with thermophysical characteristics adjusted to the surrounding environmental conditions, namely at a temperature of 28 °C. At this temperature, the air has a density (ρ) of 1.172 kg/m³ and an absolute viscosity (μ) of 1.8586 × 10^{-5} N·s/m². These characteristics are essential to guarantee the precision of the simulation in depicting vehicle operational circumstances at tropical air temperatures often observed in metropolitan settings in Indonesia or analogous climatic regions.

The CFD simulation employs a steady-state methodology utilizing the standard k-ε turbulence model, renowned for its equilibrium between predictive precision and computational efficacy in vehicle aerodynamics research. This model is able to capture the effects of turbulence quite well in the external fluid domain. The numerical solution is calculated using the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm, which is a popular iterative method for solving incompressible fluid flow equations. The SIMPLE algorithm is used to link pressure and velocity in solving the Navier-Stokes equations.

The numerical solution's accuracy is enhanced through the implementation of a second-order scheme for discretization. This approach is implemented for numerous critical variables, including pressure, momentum (x, y, and z velocities), turbulence kinetic energy (k), and turbulence dissipation rate (ε). This second-order discretization scheme allows more realistic prediction results for sharp gradients around the vehicle surface, especially in capturing boundary layer phenomena and flow separation. To ensure that the results obtained from the simulation are reliable, the convergence criterion is strictly set at a residual value of 10^{-5} for all major parameters, including continuity, velocity in three axes (x, y, and z), and turbulence parameters. This criterion aims to ensure that the simulation solution is stable and the distribution of flow variables has reached equilibrium in the fluid domain.

The boundary conditions at the inlet area are established by analyzing the variation of air inlet velocity at 10, 20, and 30 m/s. These values represent the range of vehicle operating velocity in urban environments, which range from intense traffic to open roads. Furthermore, the turbulence intensity at the inlet area is set at 5%, a standard value for external flow conditions in open environments that are influenced by obstacles from buildings and other vehicles. The boundary condition is employed as a pressure outlet on the outlet side of the domain, while the top, bottom, sides, and body of the vehicle are designated as walls (Figure 2).

In order to assess the impact of vehicle velocity variations on the drag coefficient (C_d) and lift coefficient (C_l) values, simulations were implemented in numerous velocity scenarios. The distribution of velocity, pressure, streamlines, and the numerical values of the drag coefficient and lift coefficient are the primary outputs that are observed. The interpretations were made by comparing the shape of the flow pattern, such as the wake formation behind the vehicle, and the position of the flow separation and vortex that emerge due to the body shape. The primary focus is on the size of the wake region formed behind the vehicle and the impact of the erect rear shape on the increase in drag, particularly for the square back shape.

2.2. *Grid Independency Analysis*

Table 1 presents the results of the grid independence test for C_d in four schemes, namely Mesh A to Mesh D, whose number of cells increases gradually from about 605 thousand to 908 thousand elements. The comparison shows that *Cd* decreases slightly from 0.4544 (mesh A) to 0.4536 (mesh C) with a total decrease of only 0.8 per mile, while the relative difference between meshes is always below 0.15%. Interestingly, when the mesh is refined again to mesh D, C_d increases back to 0.4542, indicating that the addition of 13% elements does not provide an increase in accuracy but instead causes small fluctuations due to numerical effects. Since the change in *Cd* between mesh B, C, and D is far below the practical threshold of 1% which is commonly used as a convergence criterion, it can be concluded that the simulation solution has achieved grid independence. With the smallest error (0.07%), mesh C is chosen in this study because it has a stable mesh and more accurate results.

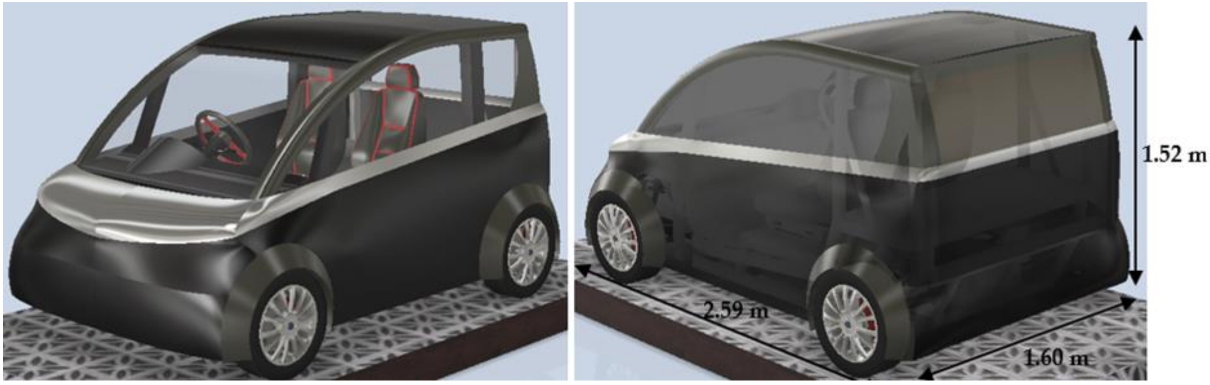

Figure 1. Two passenger city car with square back and compact shape

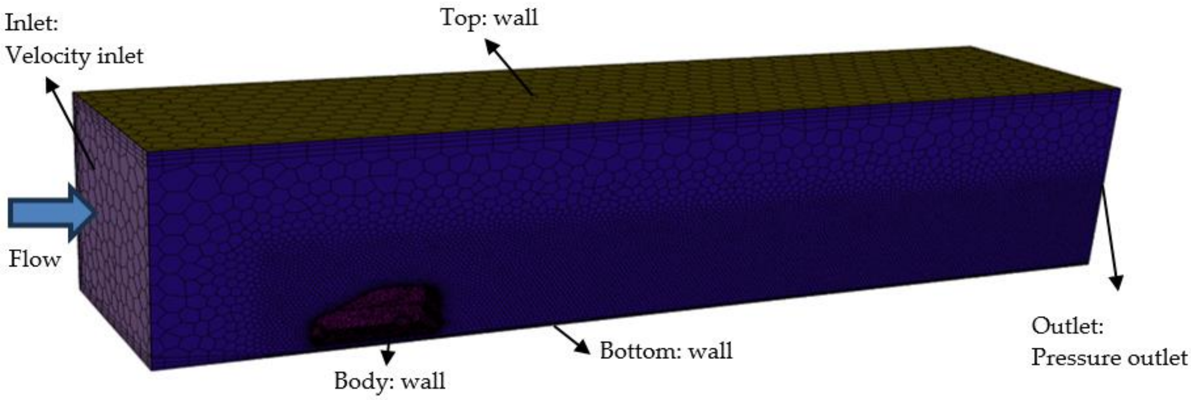

Figure 2. Boundary condition

Table 1. Analysis of grid independence

Type of Mesh	Amount of Cells	Drag Coefficient (C_d)	Error (%)
Mesh A	604638	0.4544	-
Mesh B	702547	0.4539	0.11
Mesh C	800313	0.4536	0.07
Mesh D	907734	0.4542	0.13

3. Results and Discussion

3.1. *Drag Coefficient (Cd)*

The square back-compact body vehicle model was subjected to CFD numerical simulation at three velocity variations: 10, 20, and 30 m/s. Figure 3 illustrates the trend of drag coefficient (C_d) changes. For the design of a two-passenger city car with a square-back compact body, the figure illustrates the increasing trend of C_d as the airflow velocity increases. In general, the graph illustrates a trend of increasing C_d value as velocity increases. The recorded C_d value is 0.4536 at a velocity of 10 m/s. The C_d value increases to 0.4563 as the velocity increases to 20 m/s, and it reaches 0.4581 at the maximum tested velocity of 30 m/s. This increasing trend suggests that the geometry of the vehicle generates adverse aerodynamic effects as the velocity increases, even though the absolute difference between the values is not substantial. This trend is consistent with aerodynamic theory, which posits that the wake region behind the vehicle becomes larger and flow separation occurs earlier at high velocity, thereby substantially increasing drag pressure.

The increase in C_d value is an important phenomenon to consider in the design process of two-passenger vehicles, especially those designed for mobility needs in urban areas. Although the C_d value obtained in this study is still in the moderate range for the two-passenger vehicle category, the increasing trend of C_d with increasing velocity indicates an aerodynamic challenge that cannot be ignored. This emphasizes the importance of improving the rear of the vehicle, either through passive approaches such as rounding edges or through the the integration of active aerodynamic

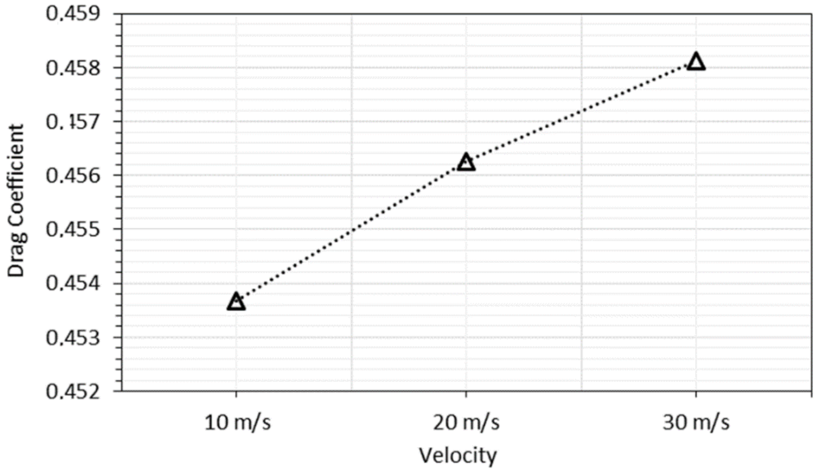

Figure 3. Drag coefficient (C_d)

technologies. In the context of two-passenger vehicles for urban use, a small increase in the C_d value can have a significant impact on the overall performance of the vehicle. This is because, at high velocity, aerodynamic drag becomes the dominant component in energy consumption, surpassing the effect of rolling resistance.

3.2. *Lift coefficient (C_l)*

Figure 4 illustrates the correlation between airflow velocity and lift coefficient (C_l) for a vehicle featuring a square rear body shape. The data presented demonstrates a progressive increase in the C_l value as velocity rises from 10 m/s to 30 m/s. At an initial velocity of 10 m/s, the concentration of C_l was measured at 0.195. The value rose to 0.212 at a velocity of 20 m/s and attained 0.222 at the maximum velocity tested, which was 30 m/s. The rise of 0.027, approximately ±14%, signifies the impact of aerodynamics on the lift force exerted on the vehicle, despite the numerical increment being quite minor.

This consistent upward trend is noteworthy, especially in the context of vehicle performance at medium to high velocity. The increase in C_l value with increasing velocity indicates that the lift force generated by the airflow over the vehicle surface is increasing. However, because the C_l value remains below the threshold of 0.25 throughout the tested velocity range, it can be concluded that this square back body design is still relatively safe and does not produce excessive lift that has the potential to reduce wheel traction and directional stability. However, there is still an opportunity for further optimization to improve overall aerodynamic performance. Several design approaches such as the addition of a diffuser to smooth the flow under the vehicle, the installation of a small spoiler to manage the rear flow, or the application of active ventilation flow technology can be considered as solutions to further suppress lift.

3.3. *Pressure Contour*

Figure 5 illustrates the pressure contours on a two-passenger vehicle under three airflow velocity conditions: 10, 20, and 30 m/s. The ANSYS simulation results indicate substantial variations in pressure distribution at each velocity, which may be quantitatively elucidated. At a velocity of 10 m/s (Figure 5a), the pressure distribution on the automobile body's surface is predominantly yellow, with pressure values varying from nearly zero to slightly positive (~0 to 134 Pa). This indicates that the drag force that occurs on the car is relatively small. When the velocity increases to 20 m/s (Figure 5b), there is a sharper decrease in pressure, especially in the front and rear corner areas of the car. This is indicated by the appearance of a green zone, with negative pressure ranging from -476 Pa to -782 Pa. This distribution signifies increased flow separation and the development of a wake region behind the vehicle body. This phenomenon results in a substantial rise in aerodynamic drag relative to the prior situation at 10 m/s.

At a velocity of 30 m/s (Figure 5c), the pressure contour shows a much wider and more intense negative pressure area, with a minimum value approaching -1851 Pa, especially at the bottom and

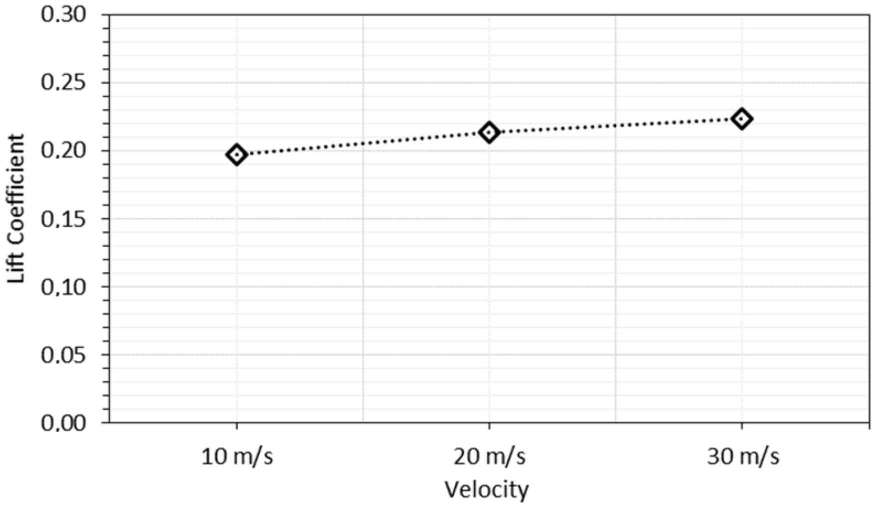

Figure 4. Lift coefficient

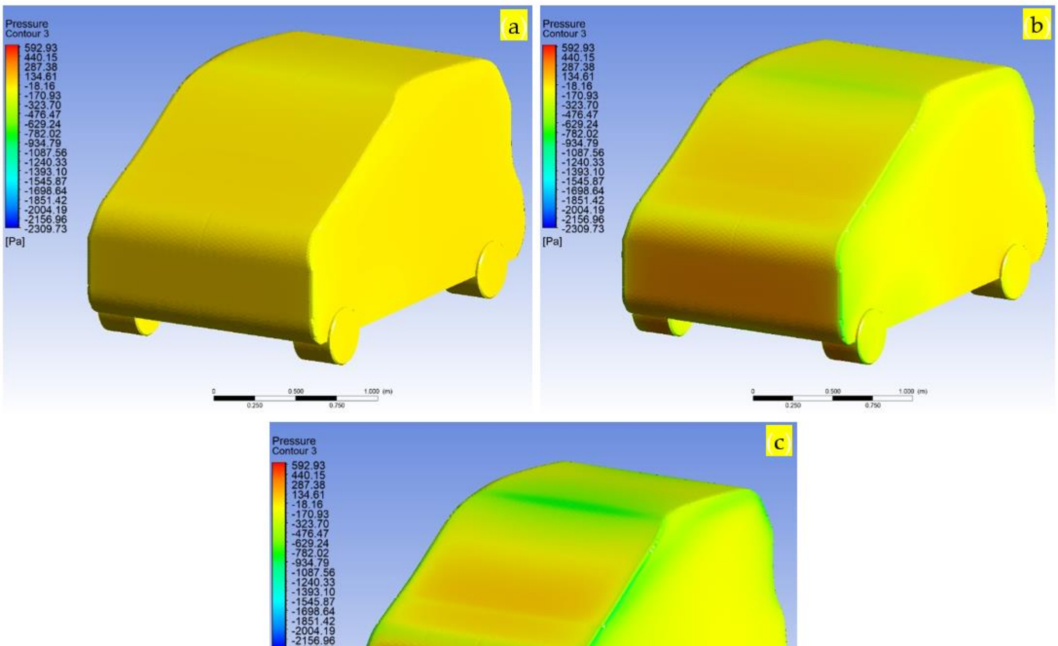

Figure 5. Pressure contour: a) 10 m/s, b) 20 m/s, and c) 30 m/s

rear of the car body. This indicates that the airflow experiences a much more extreme separation at the rear of the car. This condition causes the total drag force to increase drastically. In addition, the pressure at the front of the car also increases significantly to around 592 Pa, indicating a much stronger frontal impact of the flow.

In summary, the augmentation of airflow velocity substantially influences the pressure distribution on the automobile body's surface and directly affects aerodynamic drag. At a low velocity of 10 m/s, the pressure distribution remains rather steady with minimal drag. As the velocity escalates to 20 m/s and 30 m/s, a more pronounced pressure drop occurs, accompanied by an expansion of the negative pressure region and an increased flow separation, particularly near the vehicle's rear. At 30 m/s, the extreme negative pressure and positive pressure at the front indicate a strong frontal collision and wake, which causes a significant increase in the total drag. Thus, the aerodynamic efficiency of the car is greatly influenced by the conditions of the surrounding airflow velocity.

3.4. Velocity Contour

Figure 6 shows the contour of the airflow velocity on the horizontal plane around the vehicle body from the top view for three car velocity conditions, namely 10, 20, and 30 m/s. Generally, it is evident that an increase in vehicle velocity markedly influences the distribution and intensity of airflow, particularly in the rear of the automobile. At a velocity of 10 m/s, the airflow surrounding the car remains rather steady, as evidenced by the predominance of dark blue, indicating velocities between 3 - 10 m/s. On the side of the vehicle body, the flow is dominated by light blue to light green, indicating a velocity below 6.88 m/s. This indicates that the flow has not shown a strong turbulence pattern, and its distribution is still focused on following the contour of the car body. When the vehicle velocity increases to 20 m/s, there is a significant change in the flow pattern. From the color scale, it can be seen that the flow before hitting the vehicle reaches a velocity of close to 20 m/s, indicated by the greenish-blue color.

The airflow begins to split as it approaches the front of the vehicle and flows to the right and left sides of the body. Along the sides of the vehicle, the flow velocity decreases to about 13 m/s, indicated by the blue color. An important phenomenon occurs at the rear of the vehicle, namely the formation of a wake zone characterized by dark blue to light blue colors. In this zone, the air velocity drops drastically to below 10 m/s, indicating the occurrence of flow separation. This separation induces turbulence and low pressure behind the vehicle, which primarily contributes to aerodynamic drag due to the pressure differential between the front and rear of the car.

At the highest velocity, which is 30 m/s, the flow pattern changes more drastically. The area around the vehicle begins to be dominated by yellowish green, indicating an increase in flow velocity. However, behind the vehicle, a larger and wider wake zone is formed with a very low velocity, ranging below 6.88 m/s, as indicated by the dark blue color. This zone indicates a stronger flow separation intensity due to the upright rear body geometry, which forms a large low-pressure

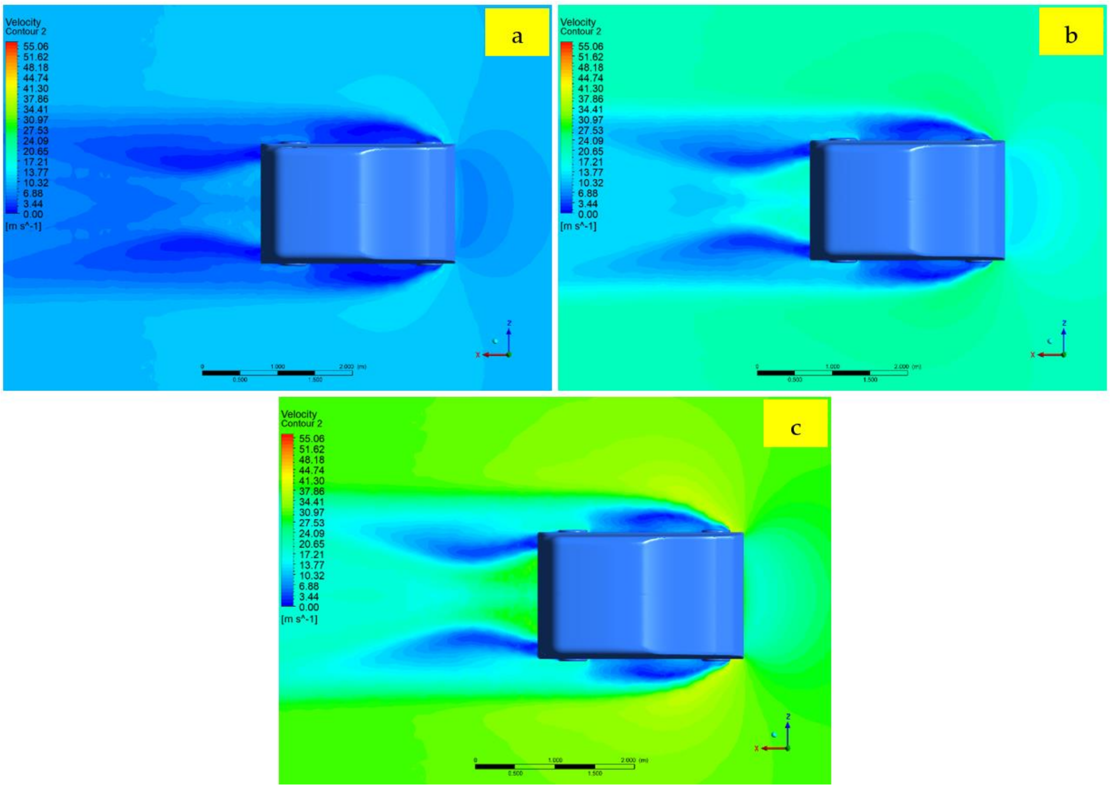

Figure 6. Velocity contour: a) 10 m/s, b) 20 m/s, and c) 30 m/s

area. Based on the horizontal scale in the image, this wake zone extends more than 1 meter from the rear of the vehicle. The larger this wake zone, the greater the aerodynamic drag experienced by the vehicle, indicating the need for a more streamlined body design to reduce drag.

3.5. *Velocity Streamline*

Figure 7 shows the airflow pattern from the rear of a two-passenger vehicle body, under conditions of airflow velocities of 10, 20, and 30 m/s respectively. This visualization illustrates the velocity streamlines, which is the trajectory of air particles as they flow past the vehicle surface, shown in the form of streamlines with a color scheme according to the velocity magnitude (blue indicates low velocity, while red indicates high velocity). From the three images, the behavior of the airflow after passing the rear surface of the vehicle can be analyzed, which is closely related to the formation of wakes (flow traces), vortices, and the potential for aerodynamic drag due to the negative pressure effect behind the vehicle.

Figure 7a shows the simulation results at an air velocity of 10 m/s. From the streamlined visualization, it can be seen that the airflow that has passed through the vehicle body experiences a significant deceleration in the rear area. A dark blue zone is formed that extends from the bottom to the top in the middle of the rear area of the vehicle, indicating that the flow velocity drops drastically due to the back pressure effect. This shows that the vehicle body creates a fairly large wake region even at relatively low velocity. The streamlines on the outside still tend to follow a straight path but begin to show slight deflection and turbulence as they approach the center of the rear of the body. This indicates that there is a flow separation from the rear surface of the vehicle which causes the flow not to follow the contour of the body and creates an eddy zone or small vortex. At a velocity of 10 m/s, although the turbulence effect is not yet too dominant, the form drag has begun to form due to the configuration of the rear shape of the vehicle which is relatively upright or bluff. Because the wake is still relatively narrow, this shows that at low velocity, the body design is not too problematic, but can still be improved to reduce the eddy effect.

Figure 7b shows the results of the flow simulation at medium velocity, which is 20 m/s. In this condition, the airflow pattern changes significantly. The wake region behind the vehicle looks larger and more complex. The blue and green zones (low to medium velocity) dominate the rear area, indicating that the airflow loses a lot of kinetic energy after passing through the body. The streamlined line shows a more turbulent pattern, with the formation of two symmetrical vortices on the left and right sides of the wake. These vortices are formed due to stronger flow release, and it is evident that the higher the velocity, the greater the inertial force of the air interacting with the rear surface of the vehicle, creating flow disturbances and earlier separation. The larger wake region has direct implications for increasing pressure drag due to the pressure difference between the front and rear of the vehicle. The formation of two large vortices at the rear of the car indicates that the body shape is not optimal in flowing air smoothly toward the rear. This can be a design input to modify the tail of the vehicle to be more tapered or use a diffuser to regulate the pressure at the bottom.

Figure 7c presents the simulation result at the highest velocity (30 m/s), showing the most extreme flow complexity. The streamlined pattern behind the body becomes more irregular and shows a massive separation phenomenon on almost the entire rear surface. The wake region enlarges with a contour that widens to the side and extends downward, indicating a very strong flow release from the vehicle body. The blue and dark green colors still dominate in the wake zone, indicating a very low flow velocity compared to the free-stream velocity of 30 m/s. This phenomenon is a strong indication that the pressure behind the vehicle is at its lowest point, causing a very large pressure deficit and drastically increasing aerodynamic drag.

3.6. *Discussion*

This research examines the aerodynamic characteristics of a two-seater square-back vehicle through a CFD simulation at three distinct velocities: 10, 20, and 30 m/s. The vehicle model exemplifies the design of a compact automobile for future urban mobility, emphasizing energy efficiency, stability, and user comfort. The simulation results show variations in the values of the drag coefficient (C_d), lift coefficient (C_l), and the distribution of pressure and velocity contours

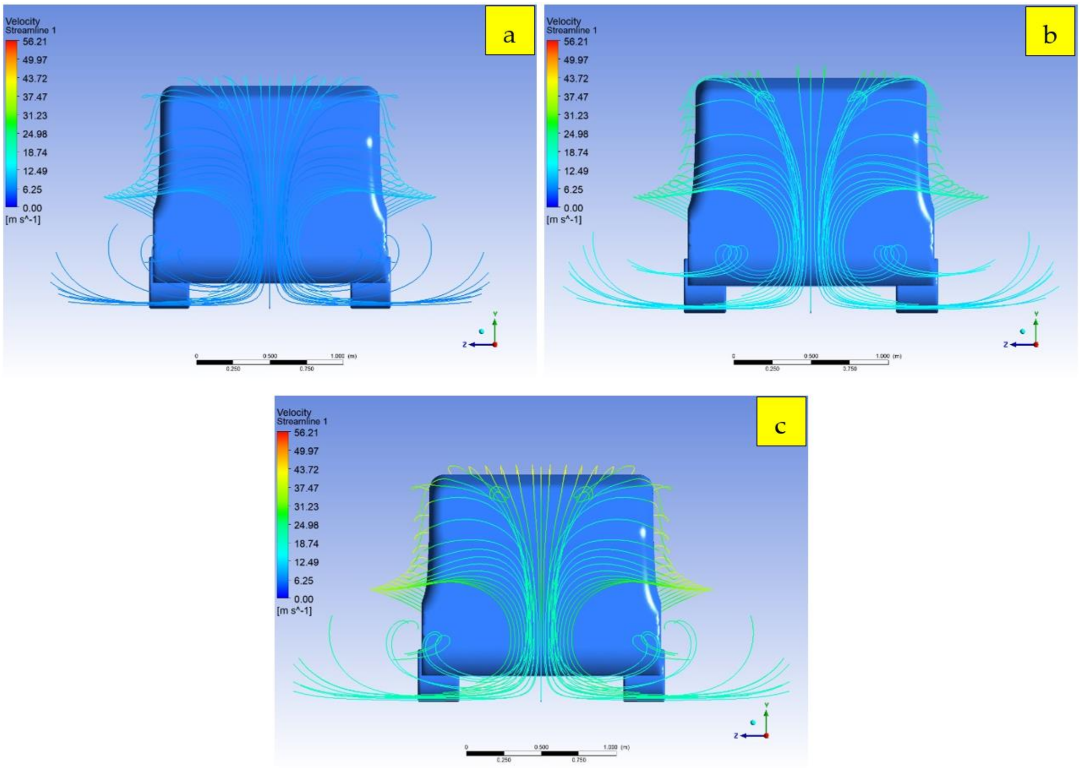

Figure 7. Velocity streamline: a) 10 m/s, b) 20 m/s, and c) 30 m/s

around the vehicle. All of these results are then further analyzed and compared with findings from previous studies.

The C_d values obtained in this study ranged from 0.4536 to 0.4581, with a tendency to increase with increasing velocity. This indicates that the square back body shape of the vehicle causes the airflow to experience significant separation at the rear, forming a large wake area and increasing drag pressure. A frequently suggested approach is to alter the geometry of the vehicle's back edge to diminish the wake size. This data aligns with prior studies indicating that wake production significantly contributes to heightened aerodynamic drag [15], [21].

On the other hand, the C_l value also shows an increasing trend, from 0.197 at a velocity of 10 m/s to 0.222 at 30 m/s. This increase in C_l indicates a tendency for the vehicle to produce greater lift at high velocity, which can hurt vehicle stability, especially in crosswind conditions or during sudden braking. To overcome this, an appropriate aerodynamic design approach is needed, such as optimizing the shape of the underbody or implementing an active diffuser. This finding is reinforced by the results of previous research which showed that light vehicles require strategies to reduce lift and drag to maintain directional stability [22].

The pressure contour distribution illustrates the concentration of high pressure at the vehicle's front resulting from flow stagnation, while low pressure prevails at the top and back of the vehicle. At a velocity of 30 m/s, the wake region behind the vehicle becomes much larger, indicating that flow separation occurs faster with increasing velocity. This phenomenon causes a decrease in pressure that directly contributes to increasing drag. This finding is consistent with research that suggests that increasing velocity increases the pressure gradient at the rear of the vehicle, which directly correlates with an increase in total aerodynamic drag [23].

The examination of velocity contours also indicated a flow pattern that aligns with prior research. Along the vehicle's side, the flow accelerates owing to the pressure differential, whilst in the back, a low-velocity zone emerges, signifying wake development. As velocity escalates, the dimensions of the wake expand, intensifying aerodynamic energy loss. This observation aligns with other research indicating that wake creation is the primary element distinguishing the aerodynamic performance of small cars from that of more streamlined conventional vehicles [24].

When compared to the classic study conducted by Ahmed et al., the Ahmed Body model with a rear angle of 35° produces a C_d value of around 0.26. Meanwhile, the C_d value obtained in this study is in a higher range. This difference is most likely due to the body shape of the vehicle in this study which is not fully streamlined, resulting in a more significant flow separation at the rear. In addition, the body surface that is less than optimal in directing airflow also increases aerodynamic drag. Nevertheless, these results still show that a slimmer and smoother body design has great potential in reducing the overall C_d value [25]. The results obtained in this study emphasize the importance of selecting a vehicle body design to optimize aerodynamic performance. In light vehicles operating in urban environments, air resistance is the main factor affecting energy consumption, especially at a velocity above 50 km/h. Therefore, reducing the C_d value is a key strategy for increasing the efficiency of light vehicles.

However, from a dynamic stability perspective, increasing lift on light vehicles is a serious challenge that must be overcome. At high velocity, increasing lift not only reduces the vertical load on the tires but can also increase the risk of aquaplaning and reduce braking effectiveness. This condition makes the vehicle more susceptible to directional and stability disturbances, especially in emergencies or bad weather. Therefore, appropriate aerodynamic engineering efforts are needed to minimize these negative effects. This finding is in line with previous research results stating that light vehicles require additional adjustments, such as the application of underbody diffusers or air dams at the front, to direct airflow and reduce lift [23], [26].

In addition, the large wake area, as seen from the velocity and pressure distributions, indicates that the vehicle is experiencing pressure drag dominance. This condition indicates that the separated airflow behind the vehicle produces a large turbulence zone that contributes significantly to the total aerodynamic drag. In such cases, the skin friction drag becomes relatively small compared to the pressure drag. Therefore, the drag reduction strategy should focus on modifying the rear body shape, rather than on treating the body surface. This finding is supported by previous research results showing that the dominance of pressure drag over skin friction drag is a common characteristic of light vehicles with vertical rear bodies [27].

4. Conclusion

The objective of this investigation is to assess the aerodynamic properties of a two-passenger vehicle with a square back body type by running a CFD simulation with a variety of airflow velocity variations. The drag coefficient (C_d) of the vehicle increases with increasing velocity, from 0.4536 at a velocity of 10 m/s to 0.4581 at a velocity of 30 m/s, as evident from the analysis results. In the same velocity range, the lift coefficient (C_l) also exhibits an increasing trend, rising from 0.197 to 0.222. The pressure contour results obtained indicate high pressure at the front of the vehicle and low pressure that is dominant at the rear. This pressure distribution causes the formation of a large wake behind the vehicle, which in turn increases the drag pressure. This condition shows that the square back body design has its aerodynamic challenges that need to be overcome. These findings confirm that the application of aerodynamic solutions such as the addition of a small rear spoiler, the use of a rear diffuser, or optimization of the rear body angle is very feasible to be further developed. This strategy is expected to significantly increase the aerodynamic efficiency of the vehicle. Overall, this study provides an important contribution to the development of aerodynamic studies for light vehicles, especially in the modern square back body design which has so far received less attention compared to streamline or fastback designs. For further research, it is recommended to conduct a more in-depth study on the effect of ground clearance, variations in rear angle (boat-tailing), and the application of active aerodynamic devices such as variable rear diffusers to reduce drag on vehicles.

Author's declaration

Authors' contributions and responsibilities

The authors made substantial contributions to the conception and design of the study. The authors took responsibility for data analysis, interpretation and discussion of results. The authors read and approved the final manuscript.

Funding

This research was funded by the Institute for Research and Community Service, Universitas Negeri Padang through a research group grant scheme with contract number: 2365/UN35.15/LT/2024.

Availability of data and materials

All data are available from the authors.

Competing interests

The authors declare no competing interest.

Additional information

No additional information from the authors.

References

[1] F. Alanazi, "Electric Vehicles: Benefits, Challenges, and Potential Solutions for Widespread Adaptation," *Applied Sciences*, vol. 13, no. 10, p. 6016, May 2023, doi: 10.3390/app13106016.

[2] A. M. Al-Ghaili, H. Kasim, H. Aris, and N. M. Al-Hada, "Can electric vehicles be an alternative for traditional fossil-fuel cars with the help of renewable energy sources towards energy sustainability achievement?," *Energy Informatics*, vol. 5, no. 4, p. 60, 2022, doi: 10.1186/s42162-022-00234-3.

[3] M. Muratori *et al.*, "The rise of electric vehicles—2020 status and future expectations," *Progress in Energy*, vol. 3, no. 2, p. 022002, Apr. 2021, doi: 10.1088/2516-1083/abe0ad.

[4] Ó. Silva, R. Cordera, E. González-González, and S. Nogués, "Environmental impacts of autonomous vehicles: A review of the scientific literature," *Science of The Total Environment*, vol. 830, p. 154615, 2022, doi: 10.1016/j.scitotenv.2022.154615.

[5] A. Ewert, M. Brost, C. Eisenmann, and S. Stieler, "Small and Light Electric Vehicles: An Analysis of Feasible Transport Impacts and Opportunities for Improved Urban Land Use," *Sustainability*, vol. 12, no. 19, p. 8098, Oct. 2020, doi: 10.3390/su12198098.

[6] R. P. Putra, D. Yuvenda, M. Setyo, A. Andrizal, and M. Martias, "Body City Car Design of Two Passengers Capacity: A Numerical Simulation Study," *Automotive Experiences*, vol. 5, no. 2, pp. 163–172, Apr. 2022, doi: 10.31603/ae.6304.

[7] R. P. Putra *et al.*, "Design and Crash Test on a Two-Passenger City Car Frame using Finite Element Method," *Automotive Experiences*, vol. 7, no. 2, pp. 270–283, Sep. 2024, doi: 10.31603/ae.11306.

[8] R. P. Putra, D. Yuvenda, W. Afnison, A. R. Saleh, and D. Harmanto, "Chassis Strength Analysis of a City Car with a Capacity of Two Passengers," in *Smart Innovation in Mechanical Engineering*, 2025, pp. 421–428. doi: 10.1007/978-981-97-7898-0_47.

[9] A. Taleghani and F. Torabi, "Editorial: Recent developments in aerodynamics," *Frontiers in Mechanical Engineering*, vol. 10, p. 1537383, Jan. 2025, doi: 10.3389/fmech.2024.1537383.

[10] C. Romero, P. Correa, E. Ariza Echeverri, and D. Vergara, "Strategies for Reducing Automobile Fuel Consumption," *Applied Sciences*, vol. 14, p. 910, Jan. 2024, doi: 10.3390/app14020910.

[11] J. Piechna, "A Review of Active Aerodynamic Systems for Road Vehicles," *Energies*, vol. 14, no. 23, p. 7887, Nov. 2021, doi: 10.3390/en14237887.

[12] H. Mansour, R. Afify, and O. Kassem, "Three-Dimensional Simulation of New Car Profile," *Fluids*, vol. 6, no. 1, p. 8, Dec. 2020, doi: 10.3390/fluids6010008.

[13] J. Waluyo, S. Hariyanto, and N. Lestari, "Evaluation of Spoiler Design on the Aerodynamic Performance of an Energy-Efficient Car Prototype Body," *Engineering and Technology Journal*, vol. 09, pp. 4972–4977, Aug. 2024, doi: 10.47191/etj/v9i08.49.

[14] N. Kamal *et al.*, "A Review of Aerodynamics Influence on Various Car Model Geometry through CFD Techniques," *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*, vol. 88, pp. 109–125, Oct. 2021, doi: 10.37934/arfmts.88.1.109125.

[15] R. D. R. Mariaprakasam, S. Mat, P. M. Samin, N. Othman, M. Ab Wahid, and M. Said, "Review on flow controls for vehicles aerodynamic drag reduction," *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*, vol. 101, no. 1, pp. 11–36, 2023.

[16] R. Stephen, "Research and Development of a Body Kit for Passenger Cars to Enhance

Aerodynamic Efficiency," Kaunas University of Technology, 2021.

[17] M. G. Connolly, A. Ivankovic, and M. J. O'Rourke, "Drag reduction technology and devices for road vehicles - A comprehensive review," *Heliyon*, vol. 10, no. 13, p. e33757, 2024, doi: 10.1016/j.heliyon.2024.e33757.

[18] F. Szodrai, "Quantitative Analysis of Drag Reduction Methods for Blunt Shaped Automobiles," *Applied Sciences*, vol. 10, no. 12, p. 4313, Jun. 2020, doi: 10.3390/app10124313.

[19] G. Sivaraj, K. M. Parammasivam, M. S. Prasath, P. Vadivelu, and D. Lakshmanan, "Low analysis of rear end body shape of the vehicle for better aerodynamic performance," *Materials Today: Proceedings*, vol. 47, no. xxxx, pp. 2175–2181, 2021, doi: 10.1016/j.matpr.2021.05.521.

[20] M. Aultman, "Computational Investigation of Incremental Design Changes on the Aerodynamics of a Production Automotive Vehicle: Roof Spoiler and Underbody Geometry," The Ohio State University, 2023.

[21] A. Guerrero and R. Castilla, "Aerodynamic Study of the Wake Effects on a Formula 1 Car," *Energies*, vol. 13, no. 19, p. 5183, Oct. 2020, doi: 10.3390/en13195183.

[22] H. Bagheri-Esfeh, M. Rostamzadeh-Renani, R. Rostamzadeh-Renani, and H. Safihkani, "Multi-objective optimisation of drag and lift coefficients of a car integrated with canards," *International Journal of Computational Fluid Dynamics*, vol. 34, no. 5, pp. 346–362, May 2020, doi: 10.1080/10618562.2020.1766031.

[23] D. S. Nath, P. C. Pujari, A. Jain, and V. Rastogi, "Drag reduction by application of aerodynamic devices in a race car," *Advances in Aerodynamics*, vol. 3, no. 1, p. 4, Dec. 2021, doi: 10.1186/s42774-020-00054-7.

[24] G. Shankar, G. Devaradjane, and S. Sunil, "Investigation on aerodynamic behaviour of a SUV car model with vortex generators at different yaw conditions," *Journal of Applied Fluid Mechanics*, vol. 12, no. 1, pp. 103–117, 2019, doi: 10.29252/jafm.75.253.28851.

[25] N. A. Siddiqui and M. A. Chaab, "A simple passive device for the drag reduction of an ahmed body," *Journal of Applied Fluid Mechanics*, vol. 14, no. 1, pp. 147–164, 2020, doi: 10.47176/jafm.14.01.31791.

[26] Z. Arifin *et al.*, "Aerodynamic Characteristics of Ahmed Body with Inverted Airfoil Eppler 423 and Gurney Flap on Fastback Car," *Automotive Experiences*, vol. 5, no. 3, pp. 355–370, 2022, doi: 10.31603/ae.7067.

[27] M. N. Sudin, M. Abdullah, S. Shamsudin, F. Ramli, and M. Tahir, "Review of research on vehicles aerodynamic drag reduction methods," *International Journal of Mechanical and Mechatronics Engineering*, vol. 14, pp. 35–47, Apr. 2014.