

## Effect of Various Canopy Shapes on the Drag Coefficient of Pickup Trucks

Khairil Anwar\*, Muhammad Syaiful Fadly, Muhammad Wahyu Hermanto

Department of Mechanical Engineering, Tadulako University, Soekarno Hatta Street, Palu, 94119, Indonesia

\*Corresponding author: khairilanwaruntad@gmail.com

Article history:

Received: 22 March 2025 / Received in revised form: 14 June 2025 / Accepted: 21 June 2025

Available online 1 July 2025

### ABSTRACT

The aerodynamic performance of light commercial vehicles, such as the Suzuki Carry, plays a crucial role in their fuel economy and road stability. One typical add-on, a canopy, often changes that airflow and, as a result, alters the drag acting on the vehicle. In this study, three different canopy shapes, flat, curved, and triangular, were examined to understand how each one affects the drag coefficient (Cd). To investigate this, both wind tunnel trials and CFD runs were conducted to track the airflow and measure any changes in drag with greater detail. For reference, the exact vehicle without a canopy was used as the base for comparison. From what has been observed, it is clear that adding a canopy tends to increase drag compared to leaving the cargo bed open. Of the three shapes tested, the flat canopy proved to be the most effective in increasing Cd, especially at moderate speeds. At around 80 km/h, for example, it pushed drag up by just over 11.063%. On the other hand, the curved canopy yielded the best result, adding only about 2.071% at 60 km/h. Flow images from the CFD runs showed that the flat and triangular designs disrupted the airflow more significantly, resulting in greater flow separation and larger wakes behind the truck. In contrast, the curved canopy seemed to keep the air closer to the surface, leaving less turbulence in its wake.

Copyright © 2025. Journal of Mechanical Engineering Science and Technology.

**Keywords:** Canopy, CFD, drag coefficient, pickup trucks, wind tunnel.

## I. Introduction

Pickup trucks are commonly utilized in the logistics sector due to their open cargo beds, which allow for flexible transportation of various goods. However, managing fuel efficiency remains a critical concern for operators. While drivetrain efficiency and load management are often prioritized, the influence of aerodynamic components, such as canopies, is sometimes underestimated. The shape of a canopy plays a significant role in determining how air flows around the vehicle, ultimately affecting the level of aerodynamic drag.

A better understanding of the aerodynamic implications of canopy geometry can help both fleet managers and vehicle manufacturers make informed design decisions that reduce drag. Lower drag reduces the workload on the engine, leading to decreased fuel consumption, lower emissions, and improved driving comfort due to more stable airflow around the vehicle body. Since aerodynamic drag significantly contributes to fuel consumption, especially at higher speeds, refining the external shape of vehicles is a practical and proven method to improve fuel economy [1], [2]. Research has consistently shown that lowering the drag coefficient (Cd) is one of the most effective strategies for improving fuel efficiency [3]. As vehicles with lower Cd values require less energy to counter air resistance [4], this approach is particularly beneficial.



Alshqirate *et al.* [5] conducted an aerodynamic study using OpenFOAM and the  $k-\omega$  turbulence model to analyze pickup truck behaviour within a speed range of 40–140 km/h. Their findings revealed that enclosing the cargo area reduced drag by 5.2% with a horizontal cover and up to 13% with an inclined one, primarily due to the reduction in airflow recirculation. However, the inclined design also resulted in increased lift force, which could negatively impact vehicle traction and safety. Airflow behaviour in pickup trucks has been the subject of multiple studies. Al-Garni *et al.* [6] employed experimental methods to analyze the airflow at the rear of a pickup truck, identifying a recirculation zone within the cargo bed but not above the tailgate. Similarly, Mukda [7] evaluated six different pickup truck designs using both experimental and CFD approaches. The results showed that sedan-shaped models had the lowest drag coefficients, while SUV-like designs exhibited the highest drag coefficients. Cabin and cargo bed design are also crucial factors influencing aerodynamic performance. Ait *et al.* [8] explored the impact of adding a protrusion at the rear of the cabin and observed measurable changes in drag. Markina *et al.* [9] fitted a large truck with cabin fairings and side extenders, which resulted in reduced aerodynamic resistance. Their findings showed fuel savings of 4.4% at 100 km/h and 9.31% at 90 km/h. Meanwhile, Kim *et al.* [10] investigated a tapered cab roof design that altered the airflow separation point, thereby reducing overall drag.

Chowdhury *et al.* [11] compared various deflector designs for their effectiveness in minimizing drag. Lietz *et al.* [12] investigated the contribution of side mirrors, underbody shields, and spoilers to the drag coefficient in sedans, SUVs, and trucks. Paul *et al.* [13] combined experiments and CFD simulations to assess the use of vortex generators and trunk spoilers in reducing drag in passenger cars. Their research identified specific configurations that effectively minimized aerodynamic resistance. In the context of motorcycle aerodynamics, Hanifudin *et al.* [14] used CFD simulations to examine the effect of slipstreaming. The study revealed that a trailing motorcycle experiences a considerable reduction in both drag and downforce due to the low-pressure wake generated by the lead bike. However, it also found that turbulence in the wake flow can influence the stability of the following rider.

The high market volume and demand for aftermarket accessories, such as canopies, demonstrate the strong economic potential for aerodynamic optimization. Furthermore, the choice of canopy design is discussed from an aerodynamic perspective in terms of its long-term benefits in fuel efficiency and emission reduction and its support for economic and environmental sustainability. Various studies have been conducted to analyze the drag coefficient of pickup trucks [15]-[18]. However, research specifically examining the effect of canopy shape on the drag coefficient remains limited. Therefore, studying the influence of canopy shape on drag coefficient is crucial for optimizing the aerodynamic performance of pickup trucks. This study specifically focuses on the Suzuki Carry pickup truck, comparing three canopy shape variations, curved, triangular, and flat, against the baseline condition without a canopy.

## II. Material and Methods

This research employs both experimental and simulation-based methods. The independent variables include canopy shape (flat, curved, and triangular) and speed (40, 60, 80, and 100 km/h), while the dependent variable is the drag coefficient ( $C_d$ ). The Taguchi approach assessed whether the differences in drag coefficients for various canopy designs were statistically significant. The model used in this study is a pickup truck based on the geometry of the Suzuki Carry. The model design was created using SolidWorks 2023,

considering the vehicles original dimensions. This model serves as a reference for analyzing airflow patterns around the vehicle and the drag coefficient when equipped with a canopy. Figure 1 presents the pickup truck model in its baseline condition without a canopy.

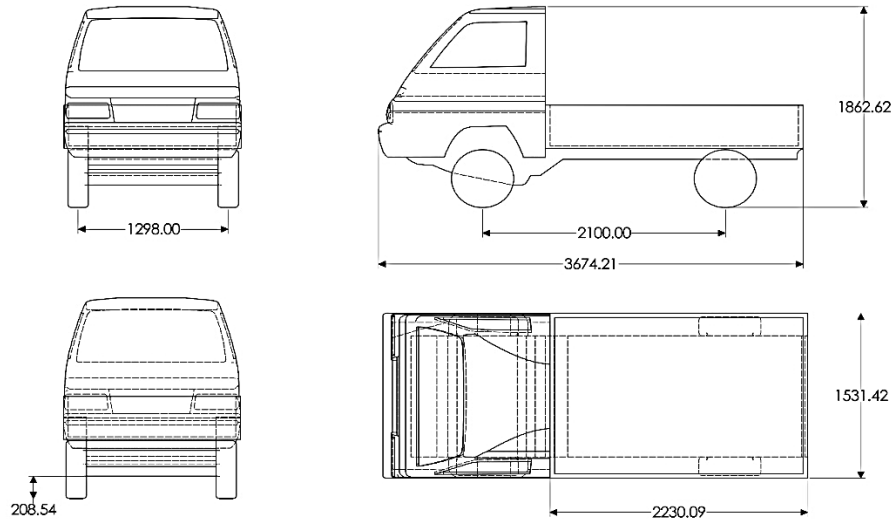


Fig. 1. Model of the pickup truck without a canopy (unit: mm)

Three types of canopies were used in this analysis, based on common modifications made by pickup truck owners frequently seen on the road. Each canopy is supported by poles on each side for structural stability and is designed to match the vehicle's width and length. Figure 2 illustrates the three canopy shapes used in this study: a flat canopy, a curved canopy resembling a tunnel tent, and a triangular canopy.

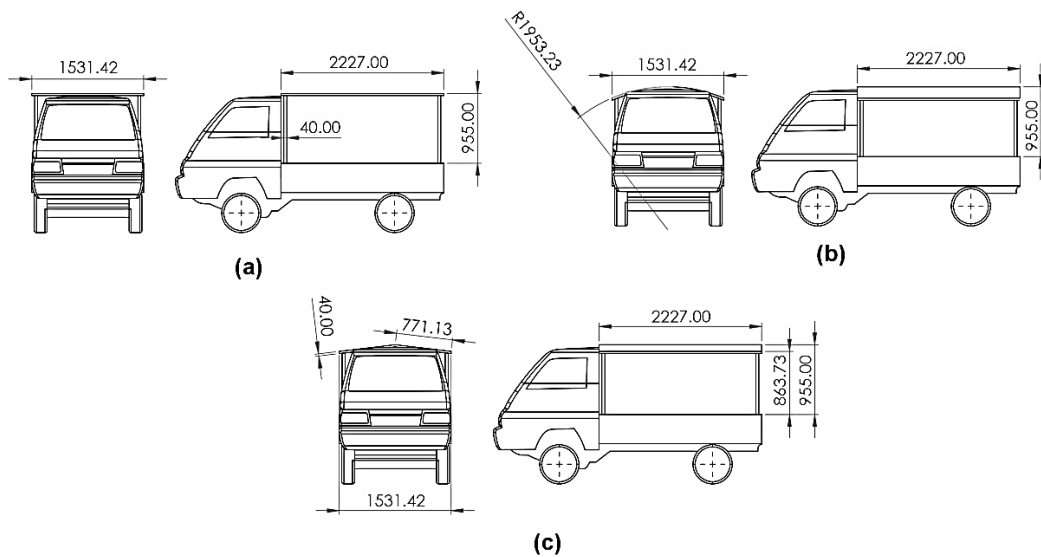


Fig. 2. Canopy models on the pickup truck: (a) Flat, (b) Curved, (c) Triangle

A wind tunnel was utilized to assess the aerodynamic drag of the experimental model. The wind tunnel was used to test the canopy shapes installed on the pickup truck model. The research object was placed in the center of the wind tunnel, as shown in Figure 3, while air was directed using an axial fan at different speeds of 40, 60, 80, and 100 km/h. The canopies with various designs were 3D-printed using PLA filament material.

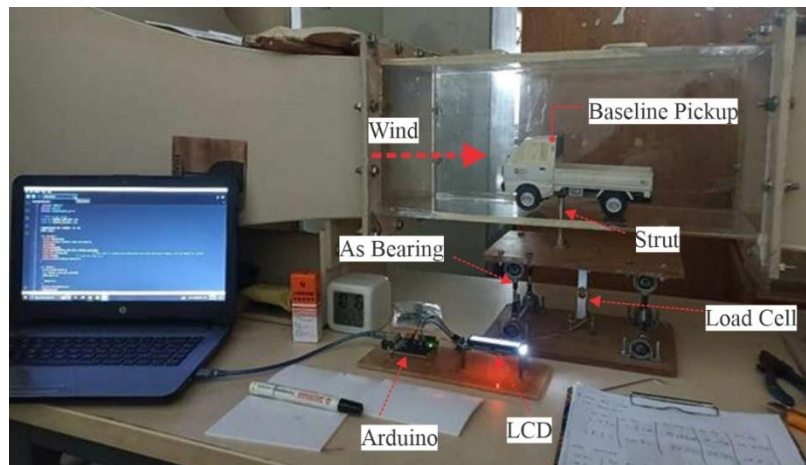


Fig. 3. Schematic of the experimental setup

The drag force measurements were conducted using a measuring device placed beneath the test apparatus in the wind tunnel. A load cell connected to the vehicle model was used to detect the forces generated by airflow in the test section of the wind tunnel. The canopies installed on the vehicle model were tested under varying airflow speeds. The load cell mounted on the test stand recorded the detected force and transmitted the data to an Arduino system. The data collected from the load cell was then displayed on an LCD screen after undergoing a calibration process.

Aerodynamic drag primarily consists of three main forces: frontal pressure, rear vacuum, and boundary layer effects [19], [20]. The combination of these forces represents the majority of airflow interactions with the vehicle body. The equation for the aerodynamic drag coefficient ( $C_d$ ) is shown in Eq. 1 [21]-[23], where  $F_d$  represents the drag force,  $\rho$  is the fluid density,  $V^2$  is the relative flow velocity, and  $A$  is the frontal area.

$$C_d = \frac{F_d}{\frac{1}{2} \rho V^2 A} \dots\dots\dots(1)$$

The CFD simulation method was used for 3D modeling. The simulation was conducted using ANSYS software. The fluid domain was created before the meshing process. This domain, also known as an enclosure, is used in the simulation analysis to represent the fluid flow around the object [24]. The dimensions of the enclosure are shown in Figure 4.

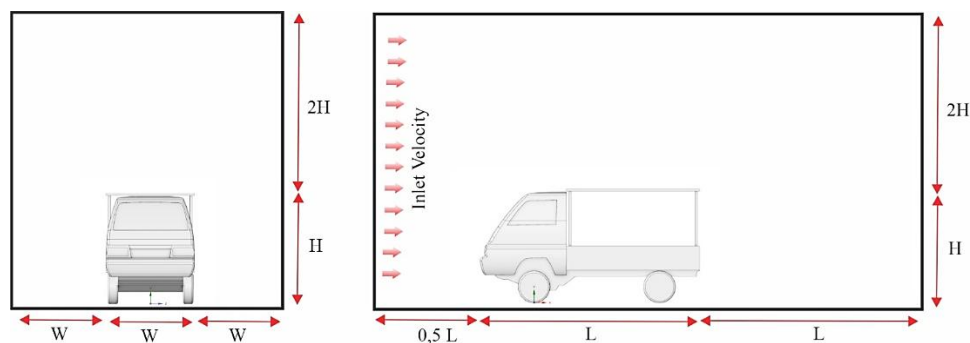


Fig. 4. The enclosure of the pickup model

The next step is the meshing process, which involves discretizing the model into a number of elements. Meshing is the most critical stage in the simulation, as it directly affects the accuracy of the results. The quality of the mesh determines the precision of the simulation outcomes. In this project, a tetrahedral mesh was used, as shown in Figure 5. During the meshing process, the positions of the inlet, outlet, and model walls were defined to ensure the simulation runs smoothly.

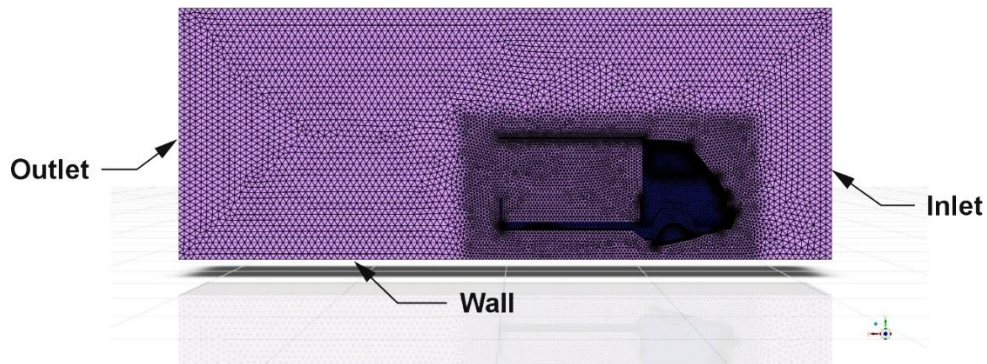


Fig. 5. Computational mesh

Meshing was performed with a local minimum size of 0.005 m and a maximum size of 0.3 m. A refinement zone was added, as the area around the vehicle is one of the primary focuses of observation. The Body-of-Influence control type was used to adjust the mesh size within the refinement zone, with a target size of 0.05 m. More detailed parameter settings can be seen in Table 1.

Table 1. CFD Setup parameter

Parameter	Specification
Pressure	1 atm
Operating Temperature	298 K (equivalent to 25°C)
Air Density	1.225 kg/m <sup>3</sup>
Viscosity (Dynamic)	1.86 x 10 <sup>-5</sup>
Mesh Configuration	Tetrahedron
Turbulence Model	Shear Stress Transport (SST) based on k- $\omega$ formulation

### III. Results and Discussions

The drag coefficient ( $C_d$ ) is an important aerodynamic parameter that determines the air resistance experienced by a vehicle. In this study, the Suzuki Carry was tested with various canopy shapes, flat, curved, and triangular, which were compared to the baseline model without a canopy to determine the drag coefficient values, as shown in Figure 6. Based on Figure 6, it can be observed that the  $C_d$  value of the baseline model without a canopy is lower compared to models equipped with a canopy. The addition of a canopy generally increases the  $C_d$  value, indicating that an additional structure on top of the vehicle increases air resistance. Among the three canopy models, the flat canopy tends to produce the highest  $C_d$  value, while the curved and triangular canopies have relatively lower values. This is due to differences in airflow around the vehicle, where a more aerodynamic shape can reduce flow separation and vortex formation behind the vehicle. A more streamlined

design helps the airflow adhere more closely to the vehicle surface, thereby minimizing flow detachment and reducing wake turbulence at the rear [25], [26].

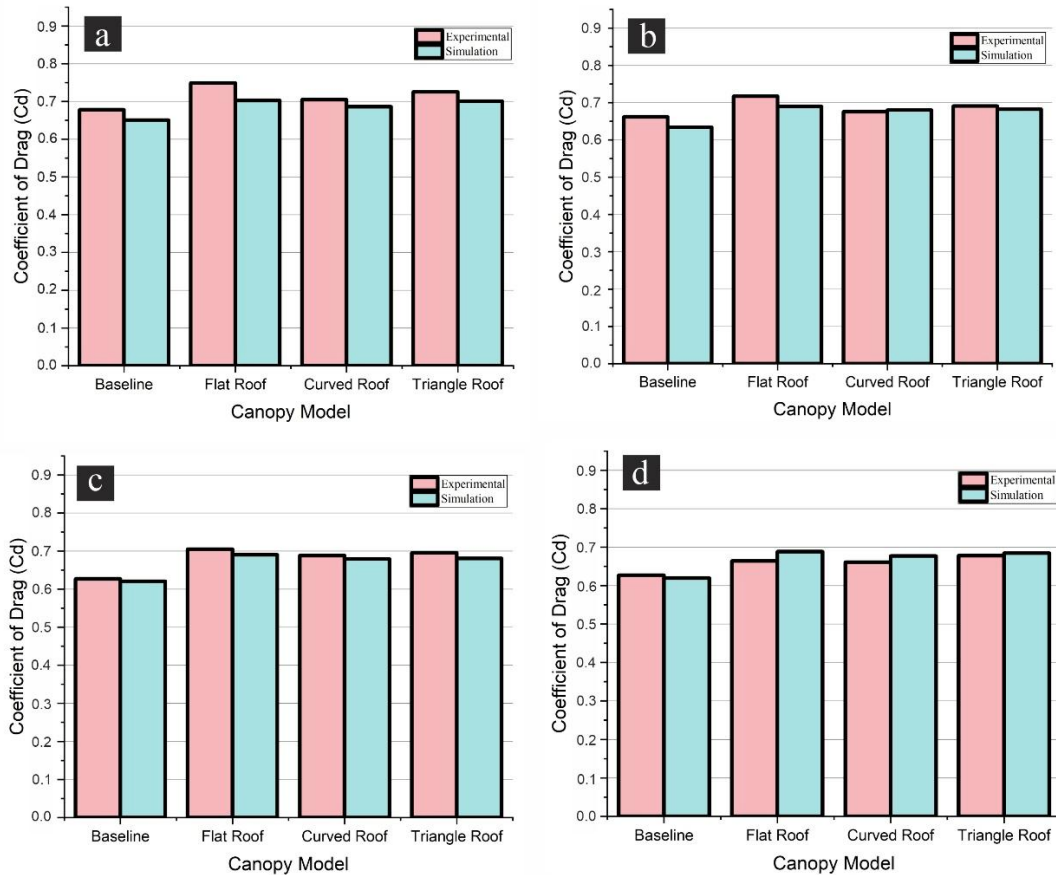


Fig. 6. Drag coefficient: (a) 40 km/h, (b) 60 km/h, (c) 80 km/h, (d) 100 km/h

Comparing the experimental findings with the CFD simulations reveals that both methods yield results that are generally in good agreement, thereby strengthening the credibility of this study. Although slight differences are inevitable, the overall pattern stays the same: when a canopy is added, the drag coefficient ( $C_d$ ) tends to rise. These minor discrepancies could be linked to real-world factors, such as wind shifts, turbulence, or minor surface variations on the vehicle, which are difficult to reproduce exactly in simulation conditions. A crucial aspect to consider is the impact of vehicle speed on aerodynamic performance. The analysis indicates that the canopy's impact on drag becomes increasingly significant as the vehicle's speed increases. This trend highlights the importance of considering typical driving speeds when selecting the most suitable canopy design. Compared to the baseline model without a canopy, vehicles fitted with a canopy show a sharper increase in drag coefficient ( $C_d$ ) at higher speeds. This observation reinforces the advantage of selecting a canopy with a streamlined profile, such as a curved shape, to minimise aerodynamic resistance under high-speed driving conditions. The aerodynamic performance of a vehicle is greatly influenced by its external design, including any added components like deflectors [27]. Dasar *et al.* [28] observed that enclosing the cargo area of a pickup truck can lower the drag coefficient, as it minimizes the recirculation zone typically present in open cargo configurations. Table 1 presents the percentage increase in drag compared to the baseline model.

Based on Table 1, the drag coefficient ( $C_d$ ) values vary depending on the canopy shape and vehicle speed. In the baseline condition without a canopy, the  $C_d$  value tends to be lower compared to models equipped with a canopy. The baseline  $C_d$  ranges from 0.678 to 0.627 in experimental testing and from 0.650 to 0.619 in simulations as speed increases from 40 km/h to 100 km/h. Models with a canopy show an increase in  $C_d$ , with the flat canopy having the highest values in almost all scenarios, especially at low to moderate speeds.

**Table 1.** Drag coefficient value and percentage increase in drag compared to the baseline model

Model	Speed (km/h)	Coefficient of Drag ( $C_d$ )		Drag increase	
		Experimental	Simulation	Experimental	Simulation
Baseline		0.678	0.650	-	-
Flat	40	0.749	0.702	9.479	7.407
Curved		0.705	0.686	3.829	5.160
Triangle		0.725	0.700	6.482	7.093
Baseline		0.662	0.633	-	-
Flat	60	0.717	0.689	7.670	8.075
Curved		0.676	0.680	2.071	6.779
Triangle		0.691	0.682	4.196	7.087
Baseline		0.627	0.620	-	-
Flat	80	0.705	0.690	11.063	10.136
Curved		0.688	0.678	8.866	8.588
Triangle		0.695	0.680	9.784	8.813
Baseline		0.627	0.619	-	-
Flat	100	0.664	0.688	5.572	10.036
Curved		0.661	0.676	5.143	8.477
Triangle		0.678	0.684	7.522	9.500

The percentage increase in the drag coefficient compared to the baseline model indicates that all canopy shapes contribute to additional aerodynamic drag, albeit in different proportions. At 40 km/h, the  $C_d$  increase due to the flat canopy reaches 9.479% in experiments and 7.407% in simulations, representing the largest increase among the models. The triangular canopy also experiences a significant increase, ranging from 6.482% to 7.093%. Meanwhile, the curved canopy shows the smallest increase, with a 3.829% rise in experiments and 5.160% in simulations. At higher speeds, this increasing trend remains evident, although with slight variations. At 80 km/h, for instance, the flat canopy exhibits a  $C_d$  increase of 11.063% (experimental) and 10.136% (simulation), while the curved and triangular canopies show slightly lower increases. Interestingly, at 100 km/h, the  $C_d$  increase tends to be lower than at 80 km/h, suggesting that airflow may become more stable at certain speeds.

Figure 7 presents the main effects plots illustrating how different canopy shapes and vehicle speeds influence the drag coefficient ( $C_d$ ). Figure 7(a) shows that the flat canopy model yields the highest average  $C_d$ , followed by the triangular and curved models. In contrast, the baseline model without any canopy exhibits the lowest drag coefficient, suggesting that adding a canopy generally increases aerodynamic drag depending on its

geometry. Meanwhile, Figure 7(b) shows that the curved canopy model achieves the highest signal-to-noise (SN) ratio, indicating greater aerodynamic stability in disturbances or variability. On the other hand, the flat canopy model has the lowest SN ratio, reflecting its weaker performance in maintaining drag consistency under changing conditions. Figure 8 presents the CFD post-processing results for velocity contours of the pickup truck model with the added canopy.

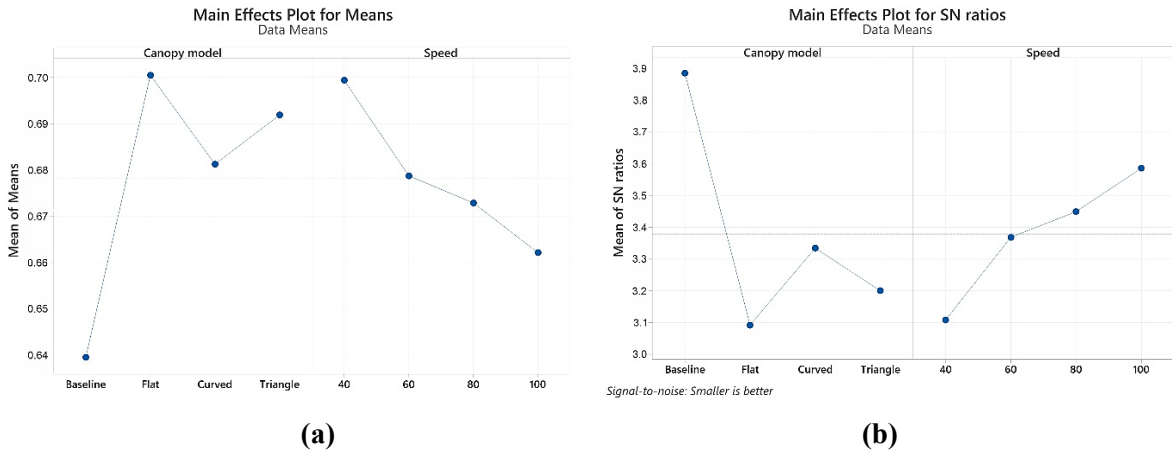


Fig. 7. (a) Main effect graphs for  $C_d$ , (b) SN ratio graphs for  $C_d$

Based on the velocity contours in Figure 8, it can be observed that the airflow pattern around the vehicle is significantly influenced by the presence and shape of the canopy. In the baseline model without a canopy, the airflow over the top of the vehicle remains relatively stable, with a smaller separation area at the rear. The green to yellow color distribution on the upper part of the vehicle indicates that the airflow velocity remains high, signifying a smoother transition without major disturbances. However, at the rear section, stagnation zones and vortices occur due to the formation of low-pressure regions, leading to aerodynamic drag, although it is still lower compared to models with a canopy. Such airflow behavior aligns with the findings of Yathiraj *et al.* [29], who emphasized that in bluff-bodied vehicles like pickup trucks, pressure drag is predominantly caused by the expansion of the wake region and the occurrence of flow separation behind the cabin. Inefficient aerodynamic features in truck bodies can also reduce directional control since disrupted roof airflow lowers lift and shifts the center of gravity [30].

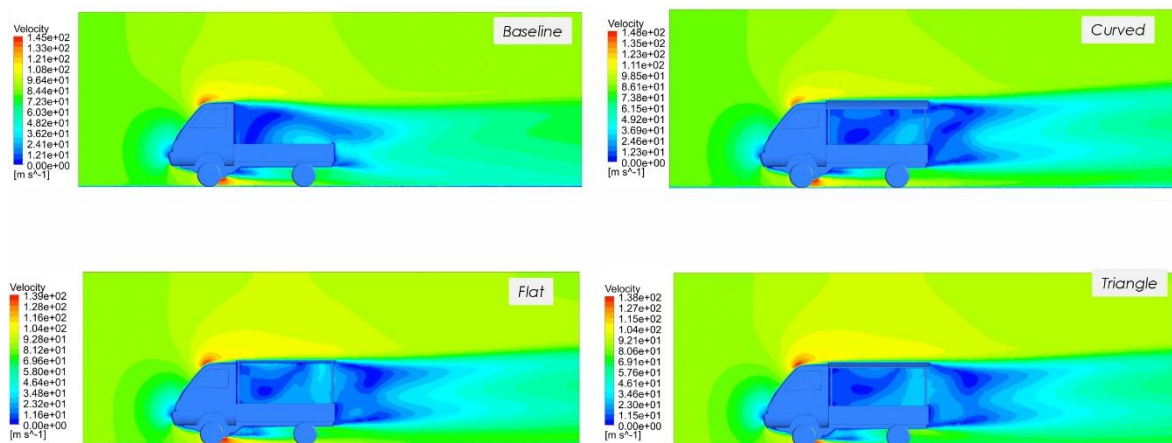


Fig. 8. Velocity contour

For models with a canopy, the changes in velocity distribution become more pronounced. The flat canopy exhibits the most significant airflow disruption, with a larger blue-colored region along the rear of the vehicle, indicating an expanded flow separation zone. This results in a higher drag coefficient compared to both the baseline model and other canopy designs. Meanwhile, the triangular canopy produces a slightly improved airflow pattern with a smoother transition at the rear, although it still has a relatively large separation region. The curved canopy demonstrates the best aerodynamic performance among the canopy models, as the airflow distribution over the top of the vehicle is more uniform, reducing flow separation and the formation of vortices at the rear. Figure 9 presents the CFD post-processing results for the velocity streamlines of the pickup truck model with the added canopy.

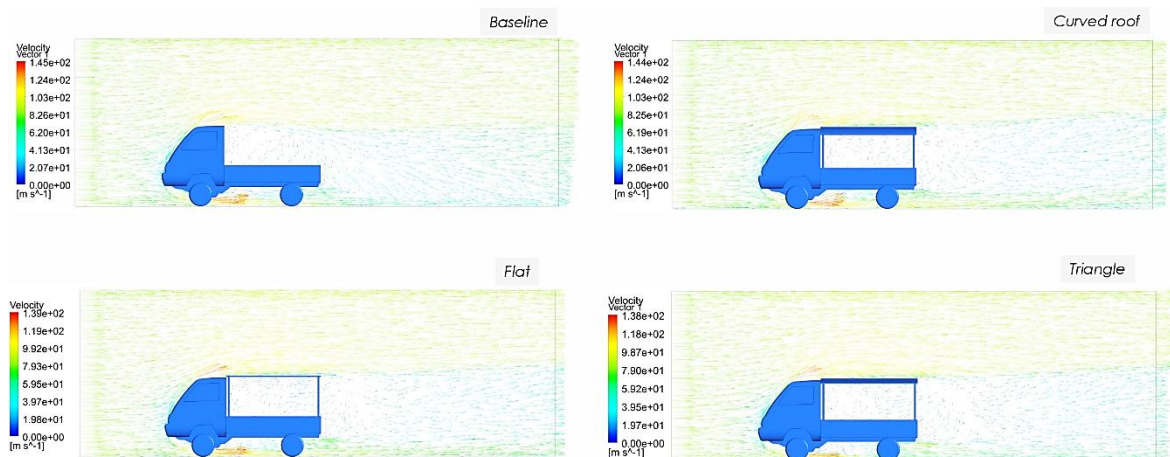


Fig. 9. Velocity streamline.

Based on the velocity vector contour analysis shown in Figure 9, the airflow distribution around the vehicle exhibits significant differences between the baseline model and the models with an added canopy. In the baseline model, the airflow over the top of the vehicle remains relatively smooth, with fewer disturbances compared to the other models. At the rear of the vehicle, a low-pressure zone is present, leading to vortex formation; however, the turbulence area is smaller than that of the models equipped with a canopy. The green-to-yellow color surrounding the vehicle indicates that air velocity remains high in most areas, signifying that aerodynamic drag is still within a lower range compared to the other models.

The analysis of airflow patterns further illustrates how canopy shape affects aerodynamics. In models fitted with a canopy, airflow disruption is more pronounced. For example, the flat canopy generates a large turbulent wake region at the rear, characterized by wider velocity variations and significant flow separation. The primary cause of this effect is the sudden change in how air shifts direction when it hits the canopy's edges, which intensifies the vortices behind the vehicle. The triangular canopy does help somewhat by reducing some of this turbulence, but it still leaves a clear area of flow separation at the rear. In contrast, when the curved canopy is used, the air follows the vehicle's shape more naturally, producing a steadier speed profile and leaving behind a smaller wake of disturbed air. This makes the curved version the most aerodynamic among those tested.

Choosing a sleeker canopy, such as the curved type, keeps the air attached longer and weakens the spinning air pockets that would otherwise trail behind the truck. On the other hand, flat or triangular shapes disrupt the flow more abruptly, creating larger separation

zones and reducing drag overall. All of these points to why the choice of canopy shape should not be an afterthought. For trucks like the Suzuki Carry, often used to transport loads day after day, a well-designed aerodynamic canopy can significantly reduce fuel consumption and provide drivers with a smoother ride, especially when travelling at higher speeds.

#### IV. Conclusions

This research investigated the effect of various canopy shapes, including flat, curved, and triangular models, on the aerodynamic drag coefficient of pickup trucks at speeds of 40, 60, 80, and 100 km/h, using both experimental testing and computational simulations. In all scenarios, the presence of a canopy led to increased drag compared to the original configuration without a canopy. Among the tested designs, the flat canopy consistently produced the highest drag values, resulting in a 9.479% increase at 40 km/h, as indicated by the experimental results. On the other hand, the curved canopy demonstrated the most efficient aerodynamic behaviour, with the lowest drag increases recorded at 3.829 % at 40 km/h and 2.071% at 60 km/h. This performance trend continued at higher speeds, where the curved model maintained its advantage in both test and simulation results. Choosing a sleeker canopy, such as the curved type, keeps the air attached longer and weakens the spinning air pockets that would otherwise trail behind the truck.

#### Acknowledgment

The authors would like to express their sincere gratitude to the Faculty of Engineering and the Department of Mechanical Engineering at Tadulako University, Palu, Indonesia, for their valuable support and contributions to this work.

#### References

- [1] C. Bayindirli, Y.E. Akansu, and M.S. Salman, "Drag reduction of truck and trailer combination with different passive flow control methods," *J. Therm. Sci. Technol. Bilim. ve Tek. Derg.*, vol. 44, no. 2, 2024, doi: 10.47480/isibted.1515727.
- [2] C.-C. Wang and C.-P. Wen, "Aerodynamic drag reduction for a truck model using DBD plasma actuators," *Adv. Mech. Eng.*, vol. 14, no. 3, p. 168, 2022, doi: 10.1177/168781322210878.
- [3] B. Basara, Z. Zunic, Z. Pavlovic, P. Sampl, A. Jemcov, W. Edelbauer, S. Saric, and U. Cvelbar, "Performance analysis of immersed boundary method for predicting external car aerodynamics," *SAE Technical Paper*, 2022. doi: 10.4271/2022-01-0889.
- [4] I.T. Ince, H. Mercan, and N. Onur, "Experimental and numerical analysis of flow over a pickup truck," *Heat Transf. Res.*, vol. 52, no. 17, 2021, doi: 10.1615/HeatTransRes.2021038655.
- [5] A.A. Alshqirate, D.Z. Ghafoor, and S.L. Borse, "CFD analysis of flow over pickup truck with and without covering cargo area using OpenFOAM," *J. Adv. Res. Fluid Mech. Therm. Sci.*, vol. 90, no. 1, pp. 40–51, 2022, doi: 10.37934/arfmts.90.1.4051.
- [6] A.M. Al-Garni, L.P. Bernal, and B. Khalighi, "Experimental investigation of the near wake of a pick-up truck," *SAE technical paper*, 2003, doi: 10.4271/2003-01-0651.
- [7] P. Mukda, "Effect from accessories on pickup aerodynamics by computational fluid dynamics," in 6th Int'l Conference on Advances in Engineering Sciences and Applied Mathematics (ICAESAM'2016), 2016, pp. 59–63, doi: 10.15242/II.E1216022.
- [8] A. Ait Moussa, J. Fischer, and R. Yadav, "Aerodynamic drag reduction for a generic truck using geometrically optimized rear cabin bumps," *J. Eng.*, vol. 2015, no. 1, p.

- 789475, 2015, doi: 10.1155/2015/789475.
- [9] A.A. Markina, A.D. Lukashuk, S.N. Chepkasov, and A.V Starovoytenko, "Improving aerodynamic characteristics for drag reduction of heavy truck," in *IOP Conference Series: Materials Science and Engineering*, 2020, vol. 862, no. 3, p. 32032, doi: 10.1088/1757-899X/862/3/032032.
- [10] J. Kim, J. Pattermann, V. Menon, and W. Mokhtar, "A CFD study of pickup truck aerodynamics," in *Proceedings of the 2011 ASEE North Central & Illinois-Indiana Section Conference*, 2011.
- [11] H. Chowdhury, B. Loganathan, I. Mustary, H. Moria, and F. Alam, "Effect of various deflectors on drag reduction for trucks," *Energy Procedia*, vol. 110, pp. 561–566, 2017, doi: 10.1016/j.egypro.2017.03.185.
- [12] R. Lietz, L. Larson, P. Bachant, J. Goldstein, R. Silveira, M. Shademan, P. Ireland, and K. Mooney, "An extensive validation of an open source based solution for automobile external aerodynamics," *SAE Technical Paper*, 2017, doi: 10.4271/2017-01-1524.
- [13] A.R. Paul, A. Jain, and F. Alam, "Drag reduction of a passenger car using flow control techniques," *Int. J. Automot. Technol.*, vol. 20, pp. 397–410, 2019, doi: 10.1007/s12239-019-0039-2.
- [14] M. Hanifudin, G. Jatisukamto, M. Darsin, M.N. Kustanto, A. Saleh, S. Prasetyono, and C.S. Sarwono, "Investigation of drag force and downforce on two racing motorcycles in slipstream conditions," *J. Mech. Eng. Sci. Technol.(JMEST)*, vol. 9, no. 1, pp. 52–61, 2025, doi: 10.17977/um016v9i12025052.
- [15] F.F. Buscariolo and R.T. Terra, "Ground height and pitch angle influence on the aerodynamic behavior of a production pickup truck," *SAE Technical Paper*, 2024, doi: 10.4271/2024-36-0011.
- [16] I.A. Ghani, R. Hassan, I. Amin, A. Suhan, K. Husni, D.H. Didane, and B. Manshoor, "Computational fluid dynamics (CFD) study on the aerodynamic of truck," *J. Des. Sustain. Environ.*, vol. 5, no. 2, pp. 23–27, 2023. <https://www.jdse.fazpublishing.com/index.php/jdse/article/view/51>
- [17] L. Fázecas, N. Avedissian, R. Ghiro, and T. Sonoda, "Improving fuel efficiency for trucks through CFD simulation," *SAE Technical Paper*, 2021, doi: 10.4271/2020-36-0410.
- [18] E. Wotango, R.B. Nallamotheu, S. Kebede, N. Bekele, A.K. Nallamotheu, and S.K. Nallamotheu, "CFD analysis on aerodynamic drag reduction of pickup vehicles using rear spoiler," in *Advances in Fluid and Thermal Engineering: Select Proceedings of FLAME 2020*, 2021, pp. 619–628, doi: 10.1007/978-981-16-0159-0\_54.
- [19] B. Song, C. Yuan, F. Permenter, N. Arechiga, and F. Ahmed, "Surrogate modeling of car drag coefficient with depth and normal renderings," in *International Design Engineering Technical Conferences and Computers and Information in Engineering Conference*, 2023, doi:10.48550/arXiv.2306.06110
- [20] 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," *J. Adv. Res. Fluid Mech. Therm. Sci.*, vol. 101, no. 1, pp. 11–36, 2023, doi: 10.37934/arfmts.101.1.1136.
- [21] N. Xavier, "Calculating the aerodynamic drag coefficient of a Toyota Avanza car CAD model using CFD analysis," in *2023 31st Southern African Universities Power Engineering Conference (SAUPEC)*, 2023, pp. 1–5, doi: 10.1109/SAUPEC57889.2023.10057807.
- [22] M.N.F. Kamal, I.A. Ishak, N. Darlis, D.S.B. Maji, S.L. Sukiman, R. Abd Rashid, and M.A. Azizul, "A review of aerodynamics influence on various car model geometry

- through CFD techniques,” *J. Adv. Res. Fluid Mech. Therm. Sci.*, vol. 88, no. 1, pp. 109–125, 2021, doi: 10.37934/arfmts.88.1.109125.
- [23] A. Backiyaraj, T. Kumaran, M. Parthasarathy, N.M. Nachippan, and P.B. Senthilkumar, “Vehicle aerodynamics and different testing methods,” in *Handbook of Research on Aspects and Applications of Incompressible and Compressible Aerodynamics*, IGI Global Scientific Publishing, 2022, pp. 388–406, doi: 10.4018/978-1-6684-4230-2.ch018.
- [24] A. Chummar and R. Harish, “CFD simulation of laminar free convection flows of nanofluids in a cubical enclosure,” *Mater. Today Proc.*, vol. 51, pp. 1473–1481, 2022, doi: 10.1016/j.matpr.2021.10.105.
- [25] S. Ganesan, R. Thiyagarajan, and P.A.K.M. Khan, “Aerodynamic performance assessment on typical SUV car model by on-road surface pressure mapping method,” *SAE Technical Paper*, 2021, doi: 10.4271/2021-28-0188.
- [26] Z. Xia and M. Huang, “Optimizing the aerodynamic efficiency of electric vehicles via streamlined design: A computational fluid dynamics approach,” *Int. J. Heat Technol.*, vol. 42, no. 3, 2024, doi: 10.18280/ijht.420315.
- [27] W. Gao, Z. Deng, and Y. He, “A comparative study of tail air-deflector designs on aerodynamic drag reduction of medium-duty trucks,” *Int. J. Veh. Perform.*, vol. 8, no. 2–3, pp. 316–333, 2022, doi: 10.1504/IJVP.2022.122138.
- [28] M. H. Dasar, D. J. Patil, and M. Raviraj, “CFD Simulation of Pickup Van,” *Int. J. Eng. Res. Technol.*, vol. 2, pp. 1995–2004. <https://www.ijert.org/research/cfd-simulation-of-pickup-van-IJERTV2IS120775.pdf>.
- [29] K. Yathiraj and P. Chaithanya, “Aerodynamics investigations and optimization of a simplified pick-up truck with wind tunnel and CFD testing,” 2024, <http://hdl.handle.net/20.500.12380/309024>.
- [30] M.S. Prasath, C. Sasikumar, and G. Sivaraj, “Numerical studies on aerodynamics performance of modified basebleed to reduce fuel consumption in squareback cars,” *J. Appl. Fluid Mech.*, vol. 18, no. 1, pp. 205–221, 2024, doi: 10.47176/jafm.18.1.2701.