AERODYNAMIC ANALYSIS OF WIND DEFLECTOR DESIGN ON PICK UP BOX CARS BASED ON CFD

Imam Krismunandar [1*], Oktarina Heriyani [2]
[1,2] *Program Studi Teknik Mesin, Universitas Muhammadiyah Prof. Dr. HAMKA, Jakarta, Indonesia*
* *Email Corresponding Author*: oktarina@uhamka.ac.id

ABSTRAK

Penelitian ini bertujuan untuk menganalisis pengaruh penambahan wind deflector terhadap performa aerodinamika mobil pick-up box melalui metode Computational Fluid Dynamics (CFD). Permasalahan utama pada kendaraan jenis ini adalah adanya hambatan tekanan (pressure drag) yang besar akibat perbedaan ketinggian antara kabin dan kargo. Simulasi dilakukan menggunakan perangkat lunak ANSYS Fluent dengan model turbulensi k-epsilon pada dua konfigurasi: model standar dan model dengan wind deflector. Hasil simulasi menunjukkan bahwa pada model standar terjadi separasi aliran yang signifikan di area celah antara kabin dan box, menghasilkan nilai koefisien hambat (C_d) sebesar 0,5239 dan gaya hambat (F_d) sebesar 2,904 N. Setelah pengaplikasian wind deflector, aliran udara menjadi lebih streamline dan titik separasi dapat diminimalisir, sehingga nilai C_d turun menjadi 0,4240 dan F_d menjadi 2,347 N. Penelitian ini menyimpulkan bahwa penggunaan wind deflector mampu memberikan reduksi gaya hambat sebesar 19,19%, yang mengindikasikan peningkatan efisiensi aerodinamika kendaraan secara signifikan.

Kata kunci: Aerodinamika, *CFD, Wind Deflector, Drag Force, ANSYS Fluent*

ABSTRACT

This study aims to analyze the effect of adding a wind deflector on the aerodynamic performance of a pick-up box truck using the Computational Fluid Dynamics (CFD) method. The primary issue with this type of vehicle is the significant pressure drag caused by the height difference between the cabin and the cargo box. Simulations were conducted using ANSYS Fluent software with the k-epsilon turbulence model on two configurations: a standard model and a model equipped with a wind deflector. The simulation results show that the standard model experiences significant flow separation in the gap between the cabin and the box, resulting in a drag coefficient (C_d) of 0.5239 and a drag force (F_d) of 2.904 N. Following the application of the wind deflector, the airflow becomes more streamlined and the separation point is minimized, reducing the C_d value to 0.4240 and F_d to 2.347 N. This study concludes that the use of a wind deflector can provide a drag force reduction of 19.19%, indicating a significant improvement in the vehicle's aerodynamic efficiency.

Keywords: Aerodynamics, CFD, Wind Deflector, Pick-Up Box, Drag Force, ANSYS Fluent

PENDAHULUAN

The increase in logistics mobility in Indonesia has driven the high use of commercial vehicles such as pickup trucks modified with the addition of cargo boxes. Technically, pickup trucks are classified as bluff bodies, or blunt objects, with complex aerodynamic characteristics. The main constraint in the design of these vehicles is the extreme geometric discontinuity between the roof of the driver's cabin and the front wall of the cargo box. This difference in height (step height) causes the vehicle to lack a streamlined shape, thus triggering significant air resistance when the vehicle is moving, especially at medium to high speeds.

Aerodynamic drag or drag force Fd is a fluid mechanics force that acts parallel to the direction of air flow but opposite to the direction of vehicle motion. This drag force is greatly influenced by the geometric shape of the vehicle body, expressed in the drag coefficient C [1]. In pickup box cars, the high Cd value is caused by the flow separation phenomenon in the cabin roof area. The released air flow cannot reattach to the box surface, thus creating a recirculation or turbulence zone in the gap between the cabin and the box. This phenomenon is confirmed by research [5] which states that the recirculation area produces low pressure (suction pressure) that pulls the vehicle backward, while on the front side of the box there is high stagnation pressure [5]. The accumulation

of this pressure difference results in pressure drag that dominates the total vehicle resistance.

Efforts to reduce drag on bluff-body vehicles can be achieved through geometry optimization with the addition of passive aerodynamic devices in the form of wind deflectors. These devices are mounted on the cabin roof to direct airflow more smoothly over the top of the cargo box. Studies using wind deflectors have been able to minimize the separation area and improve pressure distribution on the frontal surface of the cargo box [11]. By reducing turbulence in the cabin gap, the drag coefficient (Cd) can be significantly reduced. However, the performance of wind deflectors in reducing drag is highly dependent on design parameters such as the deflector's angle, height, and curvature.

In conducting aerodynamic analysis, the use of Computational Fluid Dynamics (CFD)-based simulation methods has become standard in mechanical engineering research due to its ability to visualize flow fields in detail and at a cost-effective rate compared to wind tunnel testing (Gamayel & Octavianus, 2022). Through CFD simulations, researchers can analyze flow patterns, pressure contours, and drag values numerically. This is in line with research that utilizes CFD software to identify the effect of variations in wind direction design on reducing air resistance in transport vehicles [2].

Based on the above issues, an in-depth study is needed to optimize the appropriate wind deflector design for pickup trucks. This research focuses on aerodynamic analysis using CFD-based software to compare various deflector designs. Through this research, it is hoped that quantitative data will be obtained regarding the lowest drag force and drag coefficient values, thus determining the most effective wind deflector design in improving airflow characteristics on pickup trucks. The aim is to analyze the airflow patterns formed around the pickup truck body, identify airflow separation zones in standard models and models that use wind deflectors, and determine and compare the drag coefficient values on pickup trucks through CFD simulations to determine the aerodynamic efficiency of adding a wind deflector. This research is expected to determine the magnitude of the drag force acting on the pickup truck and the percentage reduction of this force after applying the wind deflector design.

With this research, it is hoped that the results of the aerodynamic analysis of wind deflector design will provide the following benefits:

1. Provides scientific contributions regarding the characteristics of airflow in bluff-body vehicles, particularly regarding separation phenomena and wake formation in modified pickup trucks.
2. Serves as an additional reference in the application of Computational Fluid Dynamics (CFD) methods to optimize the design of commercial vehicle exterior components in an academic setting.
3. Adds to the literature on the effect of wind deflector geometry on the drag coefficient Cd, which can be used as a reference for further research.
4. Provides information on the most effective wind deflector design for minimizing air resistance, which in the long term can optimize the mechanical performance of pickup trucks at high speeds.

RESEARCH METHODS

In this study, a CFD simulation approach was used to analyze the aerodynamics of a pickup truck with the addition of a wind deflector using ANSYS Fluent software. The flowchart below shows the steps of the research process in the following sections:

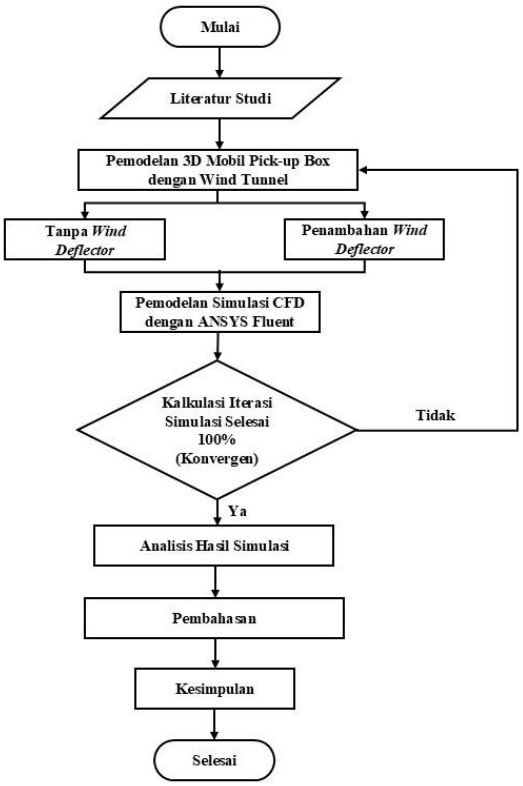


Figure 1. Research Flowchart

The initial stage of the research began with a literature review to understand the basic principles of vehicle aerodynamics, particularly the drag phenomenon in pickup box vehicles. Problem identification focused on the high air resistance caused by the height difference between the cabin and the box, which served as the basis for determining the research variables. Next, data was collected on the actual dimensions of the pickup box vehicle that would be the research object. This data included the length, width, and height of the vehicle, as well as the dimensions of the box, to ensure the model had a high level of representation of field conditions.

Based on the geometric data, a three-dimensional model was created using CAD software. The modeling was divided into two large groups: a standard (baseline) model for comparison, and a modified model equipped with various wind deflector designs. Once the CAD model was complete, the computational domain was determined, which served as a virtual wind tunnel. The next step was the discretization or meshing process, which divides the air domain into millions of small elements.

The simulation process was conducted using ANSYS Fluent software. At this stage, physical parameters were input, such as the k-epsilon turbulence model, boundary conditions, and airspeed variations. The software then performed numerical iterations until a convergence criterion was reached, at which point the residual values stabilized and no longer exhibited significant changes. The converged simulation results were then extracted in the form of quantitative and qualitative data. Quantitative data included drag force and drag coefficient (C_d). Qualitative data were presented in the form of visualizations of pressure contours, velocity vectors, and streamlines to identify flow separation points and wake zones.

The final stage involved a comparative analysis between the standard and modified models. From this comparison, conclusions were drawn regarding the effectiveness of wind deflectors in reducing aerodynamic drag and determining the most optimal design for application on pickup trucks.

The research will utilize available data on the object under study, referring to surface condenser technical documents and data obtained from process performance acquisition under actual and ideal design conditions.

The tools used in the research are a PC to perform calculations using formulas obtained from literature studies and quantities obtained from data collection on the object under study. This PC will also be used to conduct benchmark simulations using software commonly used in the heat exchanger design industry.

Research Object

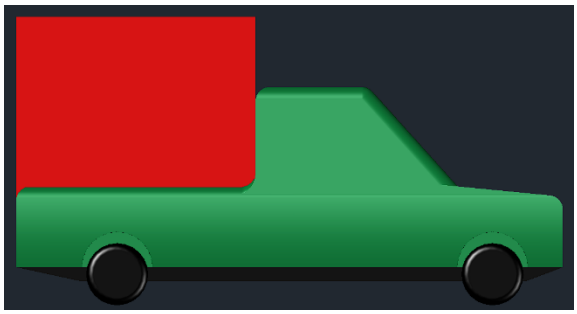


Figure 2. 3D Design of a Pick-Up Box Car Without a Wind Deflector

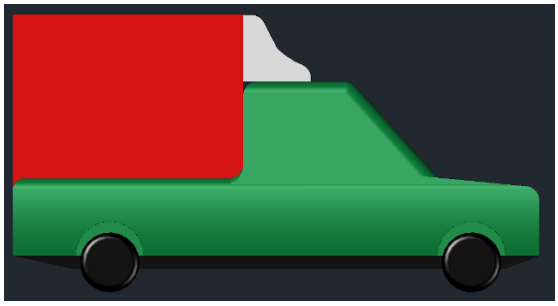


Figure 3. 3D Design of a Pick-Up Box Car with Wind Deflector

Meshing CFD

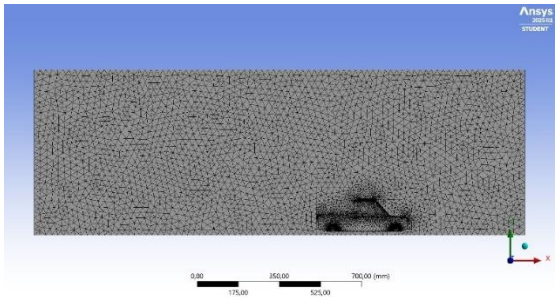


Figure 4. Meshing Results of a Pick-Up Box Car Without a Deflector

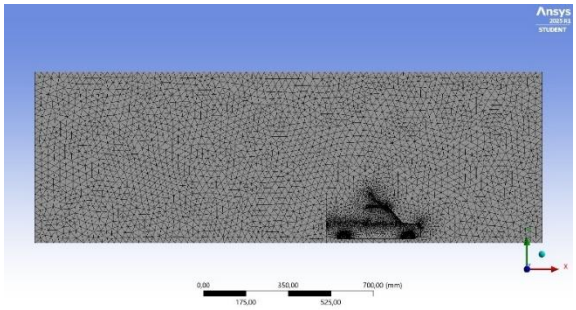


Figure 5 Results of Meshing a Pick-Up Box Car Using a Deflector

Table 1. Meshing Result Data

Pick-Up Box Car	*Element*	*Nodes*	*Avarage*	*Standard Deviation*
Without Deflector	403633	77562	0,23633	0,12928
Use a Deflector	408841	78843	0,23906	0,12928

RESULTS AND DISCUSSION

CFD Simulation Results

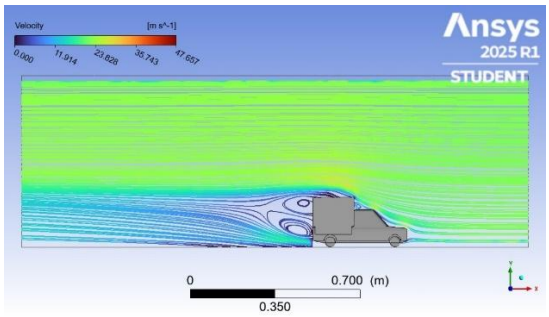


Figure 6. Airflow Pattern of a Pick-Up Box Car Without a Wind Deflector

Figure 6 shows that the airflow separates at the rear of the vehicle, forming a large wake region at low speeds. This phenomenon indicates significant aerodynamic drag due to the low pressure at the rear of the vehicle. The velocity distribution shows a maximum value of 47,657 m/s, but there are stagnation and recirculation zones that increase the drag coefficient (Cd).

In Figure 6 (without the deflector), the airflow separates significantly at the rear of the vehicle, forming a large and turbulent wake region. This region is characterized by low speeds and irregular flow, which results in low pressure behind the vehicle and high drag. This phenomenon aligns with theory [6], which states that a non-streamlined body shape will produce dominant pressure drag due to the pressure difference between the front and rear of the vehicle.

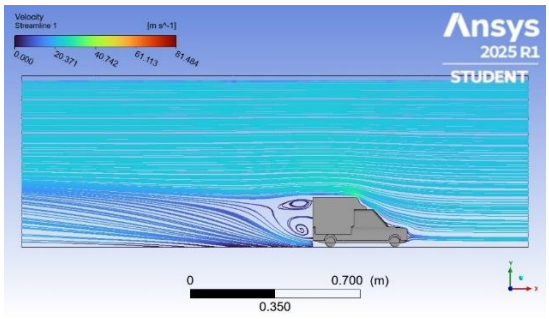


Figure 7. Airflow Pattern of a Pick-Up Box Car with Wind Deflector

Meanwhile, Figure 7 shows the simulation results with the installed wind deflector design. The airflow pattern is seen to be more directional and stable, with streamlines following the deflector contour and reducing separation at the rear of the vehicle. The maximum flow velocity increases to

81,484 m/s, but the velocity distribution in the wake area becomes more homogeneous and narrower. This indicates that the use of the wind deflector successfully reduced the turbulent area, resulting in a decrease in the Cd value.

Comparing the two figures shows that the wind deflector design has a positive impact on the vehicle's aerodynamic performance. Reducing aerodynamic drag will result in improved fuel efficiency and reduced exhaust emissions.

Discussion

Table 2 shows the CFD simulation results, including the drag coefficient (Cd) and aerodynamic drag (Fd) values for a pickup truck, both with and without a wind deflector.

Table 2. CFD Simulation Results

Pick-Up Box Car	C_d (Cd)	F_d (N)
Without Wind Deflector	0.52388869	2.9040503
Use a Wind Deflector	0.42403629	2.3468256

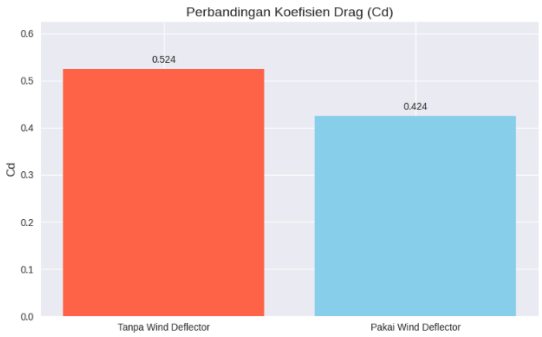


Figure 8 Bar Chart of Value (Cd) of CFD Simulation Results

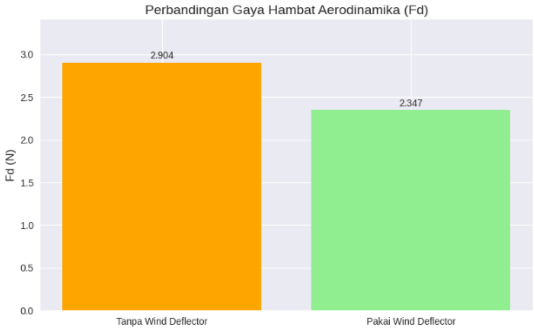


Figure 9 Bar Chart of Value (Fd) of CFD Simulation Results

The Cd value for a pickup truck without a wind deflector is 0.5239, while with a wind deflector, it drops to 0.4240. This decrease is 0.0999, or approximately 19.1%, compared to the initial condition. This indicates that installing a wind deflector significantly reduces the vehicle's aerodynamic resistance. The aerodynamic drag (Fd) for a pickup truck without a wind deflector is 2.904 N, while with a wind deflector, it drops to 2.347 N. This decrease is 0.557 N, or approximately 19.2%. This decrease in drag indicates that the deflector successfully reduces the wake area behind the vehicle, resulting in reduced negative pressure and reduced total drag.

The drag coefficient (Cd) value decreased from 0.5239 to 0.4240 after installing the wind deflector. This ±19% decrease indicates that the deflector is able to streamline the airflow pattern around the vehicle. Theoretically, a decrease in Cd means the vehicle requires less energy to overcome air resistance at a given speed. This aligns with aerodynamic theory [6], which emphasizes that vehicle body shape significantly determines drag. Aerodynamic drag (Fd) also decreased from 2,904 N to 2,347 N, or approximately 19.2%. This decrease is consistent with the decrease in Cd, as drag is directly proportional to the drag coefficient.

This drag reduction can be explained through the flow phenomena visualized in the CFD simulation. Without a deflector, the airflow experiences significant separation at the rear of the vehicle, forming a wide and turbulent wake region. With a deflector, the airflow is directed more smoothly over the top of the box, narrowing the wake region and reducing the negative pressure behind the vehicle. This results in a more even pressure distribution and reduced drag.

The 19.06% reduction in Cd in this study shows a positive and valid trend when compared to previous studies. In comparison, the study recorded a 19.37% reduction in Cd using a convex type deflector, while Kepekçi (2023) recorded a 13.38% reduction [2]. The position of the results of this study which is between the reference range (close to the results [2]) indicates that the simulated wind deflector design has functioned optimally in diverting air flow (flow redirection). This decrease in value proves that the mechanism for reducing pressure drag through eliminating stagnation pressure on the front wall of the box and reducing the wake zone at the rear has been successfully achieved through simulation.

CONCLUSION

Based on the results of Computational Fluid Dynamics (CFD) simulations and data analysis that have been carried out on the standard pick-up box car model and the model with the addition of a wind deflector, the following conclusions can be drawn.:

1. Flow Pattern and Separation Zone Analysis: Flow visualization results show significant differences in characteristics between the two models. In the standard model, a large flow separation zone was found in the gap between the cabin and the box, which triggered high turbulence. The use of a wind deflector successfully minimized this separation by directing the airflow in a streamlined manner over the top of the vehicle, so that the flow remained attached to the body.
2. The addition of a wind deflector was proven to significantly improve aerodynamic efficiency. The drag coefficient Cd decreased from 0.5239 in the standard model to 0.4240 in the model with the deflector.
3. Air drag was reduced from 2.904 N to 2.347 N. Overall, the application of this wind deflector design resulted in a drag reduction of 19.19%.

DAFTAR PUSTAKA

[1] Alawadhi, E. M. (2015). Meshing guide. In *Finite Element Simulations Using ANSYS: Vol. Release 13* (Nomor November, hal. 407–424). CRC Press. https://doi.org/10.1201/b18949-12

[2] Amadji, M., Haddad, D., & Benyahia, H. (2025). Numerical Analysis of Two Models of Air Deflectors to Improve the Aerodynamics of a Heavy Truck. *International Journal for Engineering Modelling*, *38*(2), 21–37. https://doi.org/10.31534/engmod.2025.2.ri.02f

[3] Chowdhury, H., Juwono, R., Zaid, M., Islam, R., Loganathan, B., & Alam, F. (2019). An experimental study on of the effect of various deflectors used for light trucks in Indian subcontinent. *Energy Procedia*, *160*(2018), 34–39. https://doi.org/10.1016/j.egypro.2019.02.115

[4] Gamayel, A., & Octavianus, G. (2022). *Tutorial Ansys Workbench Untuk Bidang Mekanikal* (Jilid 2). Media Sains Indonesia.

[5] Gde Didit Citra Anggarana, B., & Made Gatot Karohika, I. (2022). Analisis Aerodinamika Bodi Mobil Dengan Variasi Kecepatan Menggunakan Perangkat Lunak Cfd. *SIBATIK JOURNAL: Jurnal Ilmiah Bidang Sosial, Ekonomi, Budaya, Teknologi, dan Pendidikan*, *1*(8), 1455–1462. https://doi.org/10.54443/sibatik.v1i8.192

[6] Hucho, W.-H. (1998). Aerodynamics of Road Vehicles, Fifth Edition. In *Aerodynamics of Road Vehicles* (hal. 1–1289). SAE International. https://doi.org/10.4271/r-430

[7] KEPEKÇİ, H. (2023). Investigation of the effect of the use of top deflectors on aerodynamic performance in vehicles with CFD analysis. *International Journal of Automotive Engineering and Technologies*, *12*(2), 44–50. https://doi.org/10.18245/ijaet.1220476

[8] Slagter, W. (2020). *Accelerating Automotive CFD Simulations with High-Performance Computing*. Ansys Inc. https://www.ansys.com/blog/accelerating-automotive-cfd

[9] Sutranta, I. . (2001). *Teknologi Otomotif: Teori dan Aplikasinya*. Guna Widya.

[10] Usin, U., Fernanda, Y., Karudin, A., & Kurniawan, A. (2022). Comparison Lift and Drag of Airfoil NACA 1408 Standard and Modification Using Computational Fluid Dynamic. *MOTIVECTION : Journal of Mechanical, Electrical and Industrial Engineering*, *4*(3), 223–236.

[11] Varadarajoo, P. K., Ishak, I. A., Maji, D. S. B., Maruai, N. M., Khalid, A., & Mohamed, N. (2022). Aerodynamic Analysis on the Effects of Frontal Deflector on a Truck by using Ansys Software. *International Journal of Integrated Engineering*, *14*(6), 77–87. https://doi.org/10.30880/ijie.2022.14.06.008

[12] Versteeg, H. K., & W, M. (2007). An

Introduction to Computational Fluid Dynamics. In *Pearson Education* (Second Edi, Vol. 6, Nomor 4). Pearson Education. https://doi.org/10.1109/mcc.1998.736434