Deskripsi Artikel

- Judul Jurnal : Jurnal Asiimetrik: Jurnal Ilmiah Rekayasa Dan Inovasi
- Volume Jurnal : Volume 6, Nomor 2, 2024
- Akreditasi : Peringkat 3
- Judul Artikel : Evaluating Wind Deflector Effect on Cargo Vans Aerodynamic Drag Using Computational Fluid Dynamics
- Penulis : Agus Fikri, Riyan Ariyansah, Firman Noor Hasan, **Oktarina Heriyani**, Rosalina, Muhammad Ghiffar Sistani
- Status Penulis : Penulis ke 4

SERTIFIKAT

Direktorat Jendral Pendidikan Tinggi, Riset dan Teknologi
Kementerian Pendidikan, Kebudayaan, Riset dan Teknologi Republik Indonesia

TERAKREDITASI

Kutipan dari Keputusan Direktorat Jendral Pendidikan Tinggi, Riset, dan Teknologi
Kementerian Pendidikan, Kebudayaan, Riset dan Teknologi Republik Indonesia

Nomor: 225/E/KPT/2022

Peringkat Akreditasi Jurnal Ilmiah Periode IV Tahun 2022

Nama Jurnal Ilmiah:

Jurnal Asiimetrik: Jurnal Ilmiah Rekayasa Dan Inovasi

E-ISSN: 27162923
Universitas Pancasila
Ditetapkan Sebagai Jurnal Ilmiah:

TERAKREDITASI PERINGKAT 3

Akreditasi Berlaku selama 5 (lima) Tahun, yaitu:
Volume 4 Nomor 2 Tahun 2022 sampai Volume 9 Nomor 1 Tahun 2027
Jakarta, 7 December 2022
Plt. Direktur Jendral Pendidikan Tinggi, Riset, dan Teknologi

Prof. Ir. Nizam, M.Sc., DIC, Ph.D., IPU, ASEAN Eng
NIP. 196107061987101001

SINTA 3
SK Dirjendiktiristek, Nomor SK: 225/E/KPT/2022

p-ISSN 2655-1861
e-ISSN 2716-2923

Jurnal ASIIMETRIK

JURNAL ILMIAH REKAYASA DAN INOVASI

volume 6 nomor 2
JULI 2024

Redaksi Jurnal **ASIIMETRIK**
Fakultas Teknik Universitas Pancasila
Srengseng Sawah , Jagakarsa, Jakarta Selatan, 12640
021.789 4730 pst. 107
asiimetrik@univpancasila.ac.id

http://journal.univpancasila.ac.id/index.php/asiimetrik/

SINTA 3
SK Dirjendiktiristek, Nomor SK: 225/E/KPT/2022

p-ISSN 2655-1861
e-ISSN 2716-2923

Jurnal ASIIMETRIK

JURNAL ILMIAH REKAYASA DAN INOVASI

volume 6
nomor 2
JULI
2024

http://journal.univpancasila.ac.id/index.php/asiimetrik/

SINTA 3
SK Dirjendiktiristek, Nomor: 225/E/KPT/2022

p-ISSN 2655-1861
e-ISSN 2716-2923

Jurnal
ASIIMETRIK
JURNAL ILMIAH REKAYASA DAN INOVASI

Redaksi Jurnal ASIIMETRIK
Srengseng Sawah, Jagakarsa, Jakarta Selatan, 12640
✆ 021.789 4730 ext. 107
http://journal.univpancasila.ac.id/index.php/asiimetrik/
asiimetrik@univpancasila.ac.id

Volume 6 Nomor 2
JULI
2024

Editor-in-Chief:

- Dr. Agri Suwandi. ST., MT., CIAR., IPM.
  ID Sinta: 258280 ; ID Scopus: 56267780300

Editorial Board:

- Prof. Ir. Djoko Wahyu Karmiadji, MSME, PhD., APU.
  (*Universitas Pancasila, Indonesia*)
  ID Sinta: 600737 ; ID Scopus: 57191582540
- Prof. Dr. Ir. Dwi Rahmalina, MT.
  (*Universitas Pancasila, Indonesia*)
  ID Sinta: 5975650 ; ID Scopus: 43261707900
- Dr. Muhammad Yusro, M.T.
  (*Universitas Negeri Jakarta, Indonesia*)
  ID Sinta: 5988066 ; ID Scopus: 54974407500
- Dr. Januar Parlaungan Siregar, IPM.
  (*Universiti Malaysia Pahang Al-Sultan Abdullah, Malaysia*)
  ID Scopus: 57189757307
- Prof. Ralf Förster
  (*Berliner Hochschule für Technik (BHT), Germany*)
  ID Scopus : 8637446600
- Prof. Tim Pasang., Ph.D
  (*Werstern Michigan University's College of Engineering and Applied Sciences, USA*)
  ID Scopus : 56962784100

Section Editor:

- Ir. Duta Widhya Sasmojo, MT.
- Ari Wibowo, S.Kom., M.Kom.

Assistant Editor:

- Catur Ria Kustianti., S.Kom.
- Nurraihan Alfina Ramadhani, S.M

Jurnal Asiimetrik: Jurnal Ilmiah Rekayasa Dan Inovasi
Terakreditasi "**Peringkat 3**" berdasarkan SK Dirjendiktiristek, Nomor: 225/E/KPT/2022
Volume 6 Nomor 2 Tahun 2024, Juli 2024, Halaman 187-380
Website : http://journal.univpancasila.ac.id/index.php/asiimetrik/

p-ISSN 2655-1861
e-ISSN 2716-2923

Publisher:

- Fakultas Teknik Universitas Pancasila
  UP2M (Unit Penelitian dan Pengabdian kepada Masyarakat)

Editorial Address:

Srengseng Sawah, Jagakarsa, Jakarta Selatan, 12640
Telp. 021- 786 4730 ext. 107
asiimetrik@univpancasila.ac.id
http://journal.univpancasila.ac.id/index.php/asiimetrik/

© 2024 Jurnal Asiimetrik: Jurnal Ilmiah Rekayasa Dan Inovasi
Copyright is protected by law. It is strictly forbidden to duplicate, reproduce and reprint without written permission from the Editorial Board of Jurnal Asiimetrik: Jurnal Ilmiah Rekayasa Dan Inovasi

This work is licensed under a Creative Commons Attribution-NonCommercial 4.0 International License

SINTA 3
SK Dirjendiktiristek, Nomor: 225/E/KPT/2022
p-ISSN 2655-1861
e-ISSN 2716-2923
Jurnal ASIIMETRIK
JURNAL ILMIAH REKAYASA DAN INOVASI
REDAKSI
Volume 6 Nomor 2
JULI
2024

Jurnal Asiimetrik: Jurnal Ilmiah Rekayasa dan Inovasi is a national journal published by Faculty of Engineering Universitas Pancasila. It has been accredited "**Rank 3**" or **SINTA 3** by the Decree of the Director General of Higher Education, Research and Technology Number: 225/E/KPT/2022 and is registered with **p-ISSN 2655-1861 (print)** and **e-ISSN 2716-2923 (online)** and can be accessed via the website: http://journal.univpancasila.ac.id/index.php/asiimetrik/.

Jurnal Asiimetrik: Jurnal Ilmiah Rekayasa dan Inovasi is published regularly every **two times a year**, in **January** and **July**. This journal publishes research-based scientific articles, case studies, review articles, engineering and innovations that cover both theoretical and practical as well as their development. The topics of scientific articles published cover the fields of Architecture, Civil Engineering, Industrial Engineering, Informatics Engineering, Mechanical Engineering and Electrical Engineering.

SUMMARY. Inspired by a past design, **Santoso et al.,** were building a jaw gripper hoping to improve its efficiency by using less filament without compromising the quality of the gripper. To increase output, **Ichsan et al.**, developed an industry-wide automated system. One instrument for production simulation is the modular production system (MPS). **Setiawan et al.**, have created a vacuum gripper especially meant to handle workpieces, like labeling applications in the automation sector. The air barrier increased as **Fikri et al.**, optimized the body of the goods vehicle using CFD simulation. **Napoleon et al.**, optimized the top cover feed unit's design at the pick and place station since, upon a change in position at the time the vacuum sucks the top of the cover causes dislocation. **Allo et al**., investigated the efficiency of a hybrid drier, closely examining the material the machine dries as well as its features. With application in small-scale businesses for household appliances and fences, **Siswanto et al.** refined the TIG welding process utilizing 304 stainless steel material (SUS 304). **Dwinandana et al.** meanwhile created the ergonomic notion of a nurse-assisting robot. Using a strong and qualitative approach to ascertain the daily minimum water needs per person; **Rahman** investigated the possible needs and resources of the City of Denpasar. **Leonard et al.** worked on proposals for a child-friendly blue open playground in the seaside region of the Old Rampa Village with Bajau ethnic character while **Wijaya et al.** investigated the natural frequencies and patterns of 17-inch aluminum alloy plugs as well as the largest deformation that might occur with ANSYS software applied for simulation. **Numberi et al.** investigated the possible wind energy produced by best savonius wind turbine design. By changing the spacing between the blades and turbine beams, they conducted power tests and investigated the most ideal power for application in the coastal town of Sarmi, Papua Province. They also looked at the phenomena of heat transfer by natural convection from hot stone to food in consumer packaging. Using 6061

aluminum and ASTM B187 copper in friction welding, **Habibi et al.** carried studies aiming at estimating the strength of welding contacts. Using dandori issues, **Wibowo at al.** conducted research aiming at lowering line pauses in plastic injection operations by thirty percent. **Ridwan et al.** optimised the dehumidified air flow distribution on tray-type thermocouple dryers using CFD software. Early research on wind turbines, particularly on the efficiency of vertical axis and horizontal axis wind turbines VAWT and HAWT respectively, **Herlina et al.** The work by **Oktavian et al.** sought to ascertain the heat transfer coefficients for convection and evaporation, how the temperature of the cooling water in the condenser influences the evaporating process, and what results when the freshwater condensate level rises in a seawater desalination system. **Shafitri and Syarif** investigated developing long-range low-voltage electrical circuit breaker systems in IoT-based flood zones. Using MATLAB and microcontrollers, **Uden et al.** developed artificial neural network (ANN) testing strategies to identify voltage and current imbalances in three-phase induction engines.

SINTA 3
SK Dirjendiktiristek, Nomor: 225/E/KPT/2022
p-ISSN 2655-1861
e-ISSN 2716-2923
Jurnal ASIMETRIK
JURNAL ILMIAH REKAYASA DAN INOVASI

DAFTAR ISI

Volume 6 Nomor 2
JULI
2024

Experimental and Numerical Testing of Jaw Gripper Design Using the Mass Reduction Method of Onyx-Carbon Fiber Material at PT. Matahari Megah *Yudha Santoso*, Agus Halim, Didi Widya Utama, Kevin Raynaldo*	187-194
Design of Rejection Subsystem for Abnormal Workpiece Condition Modular Production System at Distribution Station *Christopher Adryan Ichsan*, Agustinus Purna Irawan, Agus Halim, Didi Widya Utama, William Dae Panie, Bright Levin Tolukun*	195-202
Flow Investigation Inside the Vacuum Gripper for Labeling Application with Dimensions of 100 mm × 100 mm Using the CFD Method *Eric Budiono Setiawan, Agus Halim*, Steven Darmawan, Didi Widya Utama, Kevin Raynaldo*	203-210
Evaluating Wind Deflector Effect on Cargo Vans Aerodynamic Drag Using Computational Fluid Dynamics *Agus Fikri, Riyan Ariyansah*, Firman Noor Hasan, Oktarina Heriyani, Rosalina, Muhammad Ghiffar Sistani*	211-220
Design and Optimizing Top Cover Feeding Unit Modular Production System and Pick and Place Station *Emanoelle Napoleon*, Agus Halim, Didi Widya Utama, Agustinus Purna Irawan, Jason Waworuntu*	221-226
Characteristics of Multi-tier Hybrid Dryer for Drying Corn Grains *Rombe Allo, Allo Sarira Pongsapan, David Mangallo, Agustinus, Samuel Parlindungan Siregar, Thomas Pagasis, Anastasia Sri Werdhani, Johni Jonatan Numberi, Pither Palamba, Dionisius Desriadi Banda, Joni**	227-240
Taguchi Design of Experiment (DoE) for Evaluating TIG- Welding Parameter Variations on Tensile-shear Load and Hardness Using Stainless steel 304 Material *Siswanto, Sukarman*, Dodi Mulyadi, Amir, Amri Abdulah, Ean Deka Putra*	241-252
The Development of Conceptual Design of Nurse assistance Robot's Exterior with Ergonomic Approach *Tubagus Ahmad Dwinandana*, Tota Pirdo Kasih, Muhammad Nurul Puji*	253-264
New Approach on Planning for Water Provision using Water Balance (Case Study: Sewakaderma Municipal Waterworks, Denpasar) *Herawati Zetha Rahman**	265-272
Analysis of the Children's Playground with Blue Open Space Concept in Bajau Ethnic Settlement Rampa Lama Kotabaru *Leonard, Samsu Hendra Siwi*, Titin Fatimah*	273-282

Analysis of Vibration Characteristics in 17-Inch Aluminum Alloy Wheel Rims Using Finite Element Method *Victor Indra Wijaya, Riyan Ariyansah, Delvis Agusman*, Rifky, Oktarina Heriyani*	283-294
Optimization of Savonius Turbine Performance with Variations in Blade and Shaft Spacing on the Coast of Sarmi Regency, Papua Province *Johni Jonatan Numberi*, Joni, Pither Palamba, Obed Rante Allo, Nourish C Griapon, Yane A Ansanay, Lazarus Ramandei, Wilfriedf Wanane, Rombe Allo, Yanviter Manalu, Apolo Safanpo, Endang Hartiningsi, Marthen Liga, Enos Karapa, Yosef Lefaan, Prihananto Setiadji, Herbert Innah*	295-300
Analysis of Natural Convection Heat Transfer in Barapen Cooking in Papua *Johni J. Numberi*, Pither Palamba, Agustinus Giai, Kristofel Rumar, Joni, Yane Ansanay, Obet T Ranteallo, Samuel P. Siregar, Ruben M. Kaiwai, Selyus Rantepulung, Anastasia Sri Werdani, Nourish Griapon, Yohanis Wanane, Yanviter Manalu, Apolo Safanpo, Enos Karapa, Endang Hartiningsih, Marthen Liga, Oscar O Wambrauw, Akbar Silo*	301-308
The Effect of Rotation Speed on the Quality of Friction Welding Joints in Aluminum and Copper *Habibi Santoso*, Aep Surahto, Fatimah Dian Ekawati*	309-318
Line Stop Time Reduction through Dandori Evaluation in Plastic Injection Process Production *Yohanes T. Wibowo*, Anggi A. W. Pratama, Vuko A. T. Manurung*	319-326
Optimization of Dehumidification Air Flow Distribution in Temulawak Tray Dryer with Computational Fluid Dynamics *Ridwan*, Farul Apriansa, Rudi Irawan*	327-340
Preliminary Study on Wind Turbines for Power on Floating Net Cages *Herlina*, Sri Agustina, Sariman, M. Akbar Pratama, M. Ichsan Azhary*	341-350
Convective Coefficient and Evaporative in Forced Flow Solar Still *Akbar Oktavian, Dan Mugisidi*, Rizky Alamsych, Oktarina Heriyani*	351-360
IoT Based Remote Low Voltage Power Circuit Breaker System in Flood Areas *Ainil Syafitri*, Kemil Syarif*	361-368
Diagnosing Voltage and Current Imbalance of Three-Phase Induction Motor with Artificial Neural Network Method *Suparman Uden*, Sofian Yahya, Adnan Rafi Al Tahtawi*	369-380

*Corresponding Author

Jurnal Asiimetrik: Jurnal Ilmiah Rekayasa Dan Inovasi
Terakreditasi "**Peringkat 3**" berdasarkan SK Dirjendiktiristek, Nomor: 225/E/KPT/2022
Volume 6 Nomor 2 Tahun 2024, Juli 2024, Halaman 211-220
Website : http://journal.univpancasila.ac.id/index.php/asiimetrik/

p-ISSN 2655-1861
e-ISSN 2716-2923

Evaluating Wind Deflector Effect on Cargo Vans Aerodynamic Drag Using Computational Fluid Dynamics

Evaluasi Pengaruh *Wind Deflector* terhadap Gaya Hambat Aerodinamis Mobil Box dengan Dinamika Fluida Komputasional

Agus Fikri[1], Riyan Ariyansah[1*], Firman Noor Hasan[2], Oktarina Heriyani[1], Rosalina[3], Muhammad Ghiffar Sistani[1]
[1]Program Studi Teknik Mesin, Fakultas Teknologi Industri dan Informatika, Universitas Muhammadiyah Prof. DR. HAMKA, Jl. Tanah Merdeka, No.6, Ciracas, Jakarta Timur, Indonesia
[2]Program Studi Teknik Informatika, Fakultas Teknologi Industri dan Informatika, Universitas Muhammadiyah Prof. DR. HAMKA, Jl. Tanah Merdeka, No.6, Ciracas, Jakarta Timur, Indonesia
[3]Program Studi Teknik Elektro, Fakultas Teknologi Industri dan Informatika, Universitas Muhammadiyah Prof. DR. HAMKA, Jl. Tanah Merdeka, No.6, Ciracas, Jakarta Timur, Indonesia

Article information:

Received:
04/01/2024
Revised:
04/03/2024
Accepted:
19/03/2024

Abstract

Suboptimal design and body shape in freight transport vehicles can lead to increased aerodynamic drag. To address this issue, the use of wind deflectors is proposed as a solution to reduce aerodynamic resistance in cargo vans. The methodology employed in this research involves Computational Fluid Dynamics (CFD) simulations using the Ansys Fluent R2 2023 software. CFD simulations were conducted on the design of a cargo box vehicle with variations in Wind Deflector Models 1, 2, and 3, employing identical boundary condition parameters. The results of the CFD simulation for Wind Deflector Model 3 exhibited the lowest drag force at 1.1531116 Newton and a drag coefficient of 0.37031338. In conclusion, a comprehensive analysis of the CFD simulation results provides valuable insights into the intricate aerodynamic implications of Wind Deflector variations on cargo vans. Therefore, it is concluded that Wind Deflector Model 3 emerges as the optimal choice, showcasing superior aerodynamic characteristics.

Keywords: cargo van, wind deflector, CFD simulation, drag force, drag coefficient.

***SDGs*:**

Abstrak

Desain dan bentuk bodi yang tidak optimal pada kendaraan angkutan barang dapat menyebabkan peningkatan hambatan udara. Untuk mengatasi masalah ini, penggunaan *wind deflector* diusulkan sebagai solusi untuk mengurangi hambatan aerodinamis pada van kargo. Metodologi yang digunakan dalam penelitian ini melibatkan simulasi CFD menggunakan perangkat lunak Ansys Fluent R2 2023. Simulasi CFD dilakukan pada desain mobil box kargo dengan variasi bentuk *Wind Deflector* Model 1, Model 2, dan Model 3, dengan menggunakan parameter kondisi batas yang identik. Hasil simulasi CFD *Wind Deflector* Model 3 menunjukkan gaya hambat terendah sebesar 1.1531116 Newton dan koefisien hambat sebesar 0.37031338. Sebagai kesimpulan, analisis menyeluruh terhadap hasil simulasi CFD memberikan wawasan berharga tentang implikasi aerodinamis yang rumit dari variasi *Wind Deflector* pada mobil van kargo, maka didapat kesimpulan bahwa *Wind Deflector* Model 3 muncul sebagai pilihan optimal menampilkan karakteristik aerodinamis yang unggul.

Kata Kunci: mobil box, *wind deflector*, simulasi CFD, gaya hambat, koefisien hambat.

*Correspondence Author
email : riyan_ariyansah@uhamka.ac.id

This work is licensed under a Creative Commons Attribution-NonCommercial 4.0 International License

1. INTRODUCTION

In the era of modern transportation, mobility emerges as a crucial element supporting economic growth and social comfort. The utilization of motorized vehicles, particularly in the realm of freight transportation, has surged alongside the increasing trends in trade and distribution (Paminto, 2020). Consequently, technologies related to freight transport vehicles have also undergone significant advancements. However, the design of freight transport vehicles, including cargo vans characterized by their predominantly box-like shape, remains a focal point in efforts to enhance aerodynamic efficiency. In this context, the implementation of wind deflector devices on cargo vans becomes a highly relevant factor in reducing the coefficient of drag and overall aerodynamic resistance. This, in turn, contributes to the development of cargo van designs that are more aerodynamic.

To comprehend the inherent aerodynamic challenges faced by cargo van, an essential grasp of the theory concerning drag force becomes imperative. The aerodynamic drag force acting on a vehicle is the resultant effect of air resistance encountered during the vehicle's movement through the atmospheric medium (Nath *et al.*, 2021). The deployment of Wind Deflectors emerges as a pragmatic strategy to mitigate aerodynamic drag in vehicles (Hariram *et al.*, 2021). These devices, modifiable or installable at specific sections of the vehicle, are meticulously crafted to redirect airflow, thereby ameliorating drag force comprehensively. The substantial reduction of drag force on cargo vans assumes pivotal importance in enhancing the sustainability of cargo transportation. Cargo vans, including trucks and freight-carrying vehicles, attract attention owing to their inherent aerodynamic profiles, which tend to instigate higher drag forces. The utilization of Wind Deflectors, also known as air shields, has become a prevalent strategy in endeavors aimed at refining the aerodynamic efficiency of these vehicles.

The application of CFD simulations confers a strategic advantage by enabling the virtual testing of diverse Wind Deflector configurations and other design variations on cargo vans. This capability permits the exploration of potential aerodynamic strategies without the need for resource intensive physical prototype production (Ariyansah, 2019). CFD represents a numerical approach widely employed for modeling and analyzing fluid flow around objects with intricate geometries, obviating the need for costly and intricate physical testing (Gamayel and Octavianus, 2021).

The underlying theory of CFD finds its roots in the Navier-Stokes equations a set of partial differential equations formulated to model fluid behavior (Li *et al.*, 2023). Employing discretization methods such as Finite Volume or Finite Element, the numerical solution of the Navier-Stokes equations yields comprehensive insights into pressure distribution, velocity, and other aerodynamic parameters along the surface of the vehicle (Kshirsagar and Chopade, 2018). Thus, CFD serves as a powerful tool for accurate simulations elucidating the intricate interaction between airflow and cargo vans.

The ANSYS Fluent software is widely utilized for CFD simulations in various research scenarios, particularly for analyzing the aerodynamics of vehicles. A study conducted by Yulianto, Ariyansah, and Octavianus in 2023 employed the ANSYS Fluent software version R2 2022 to investigate the influence of design angle variations on the U18C CC-201 locomotive's drag coefficient and aerodynamics (Yulianto, Ariyansah and Octavianus, 2023). The research findings elucidate that a 150-degree angle modification on the upper part of the GE U18C locomotive body has a significant impact, resulting in a notable reduction of up to 29% in the drag coefficient.

Furthermore, the study conducted by Varadarajoo et al., delves into the aerodynamic analysis of the impact of the Frontal Deflector on trucks using the ANSYS Fluent software (Varadarajoo *et al.*, 2022). The drag force generated by airflow separation at the rear of the vehicle can be mitigated through the implementation of a Frontal Deflector. Despite creating drag, this research demonstrates that the Frontal Deflector can reduce drag by preventing airflow separation at the rear of the truck. Simulations involving six different forms of Frontal Deflector reveal that Frontal Deflector 4 exhibits

the lowest drag coefficient, reaching 0.508 with a height of 15 mm and a placement of 230 mm. With a drag reduction of 32.2%, Frontal Deflector 4 is recognized as the optimal model for enhancing the aerodynamic efficiency of trucks.

The pursuit of optimizing aerodynamic efficiency in cargo vans has garnered considerable attention in prior research, particularly with an emphasis on the effectiveness of Wind Deflectors. Pevitt et al., conducted a seminal study, showcasing the successful application of CFD simulations on heavy commercial vehicles (Moriaand and Alam, 2012). This work has laid a robust methodological foundation for the comprehensive analysis of aerodynamic characteristics. Notably, this methodology facilitates detailed visual modeling of airflow patterns around the vehicle and offers adaptability to various experimental conditions within the virtual realm.

The contribution of Moriaand and Alam, work serves as a noteworthy precedent, illustrating the efficacy of CFD simulations in gaining a nuanced understanding of aerodynamic features in commercial vehicles (Moriaand and Alam, 2012). By employing this methodology, researchers can conduct virtual experiments with a high degree of accuracy, exploring diverse scenarios and configurations to optimize the design of Wind Deflectors for enhanced aerodynamic performance in cargo vans. This approach not only advances scientific knowledge in aerodynamics but also presents a cost-effective and efficient means to refine and innovate vehicle design without the necessity for extensive physical prototypes.

McTavish and McAuliffe, not only emphasize the broader potential for aerodynamic optimization in heavy-duty vehicles but also provide a contextual backdrop for endeavors aimed at enhancing fuel efficiency (McTavish and McAuliffe, 2021). This study introduces a stochastic simulation methodology to refine the accurate estimation of the efficiency of aerodynamic drag reduction technologies in ground vehicles, particularly heavy-duty vehicles. This approach utilizes specific wind conditions on designated routes to predict the speed and yaw angle of the vehicle. Predictions of aerodynamic drag and fuel consumption are grounded in the vehicle's aerodynamic performance model derived from wind tunnel measurements, road measurements, or CFD simulations.

Simulations are employed to assess the benefits of drag reduction technologies in heavy-duty vehicles on two main routes in Canada, illustrating that fuel savings can be achieved at low speeds, such as 50 km/h, with a noteworthy contribution from tractor-trailer platooning approaches. Evaluation across various vehicle weights indicates that, even as the proportion of fuel consumption attributed to aerodynamics decreases at low speeds, drag reduction continues to yield substantial fuel savings. This not only supports fuel cost reduction but also contributes to the mitigation of greenhouse gas emissions.

Despite numerous studies investigating the implementation of Wind Deflectors on motor vehicles and their impact on aerodynamic resistance, there exists a notable research gap, particularly in the context of cargo vans. Therefore, there is an urgent need to fill this knowledge gap and delve into how Wind Deflectors can be optimized to enhance aerodynamic efficiency in cargo vans, which, in turn, can significantly contribute to the sustainability of freight transportation. The novelty of this research lies in its specific focus on cargo vans and the utilization of CFD technology as the primary tool to investigate the complex aerodynamic phenomena associated with these vehicles. This study aims to analyze the effects of Wind Deflectors on the aerodynamic drag force of cargo vans. By achieving the objectives of this research, it is anticipated to provide profound insights into how the addition of Wind Deflector devices can improve aerodynamic efficiency in cargo vans.

2. METHODOLOGY

In this study, the primary methodology employed is the CFD simulation approach. However, it is imperative to emphasize that the implemented CFD simulation approach is not utilized in isolation rather, it is intricately connected to both experimental and analytical methodologies.

This synergistic integration is strategically devised to address potential errors and deficiencies in parameter data during the execution of CFD simulations, with the overarching goal of mitigating, if not eliminating, uncertainties inherent in the research process (Tu *et al.*, 2023).

This strategic integration unfolds avenues for deriving comprehensive advantages from the amalgamation of empirically verifiable experimental data, in-depth mathematical analyses, and advanced numerical simulations. Consequently, this methodological framework establishes a robust foundation, thereby enhancing the reliability and credibility of the research outcomes. The amalgamation of these three approaches provides the requisite flexibility to delve into specific facets that may pose challenges if approached solely through one method. Therefore, the success of this research is contingent not solely upon the excellence of the CFD simulation approach but also on the harmonious and synergistic integration with experimental and analytical methodologies.

2.1. Research Flow

The progression of this investigation is elucidated in the research flowchart as shown in Figure 1.

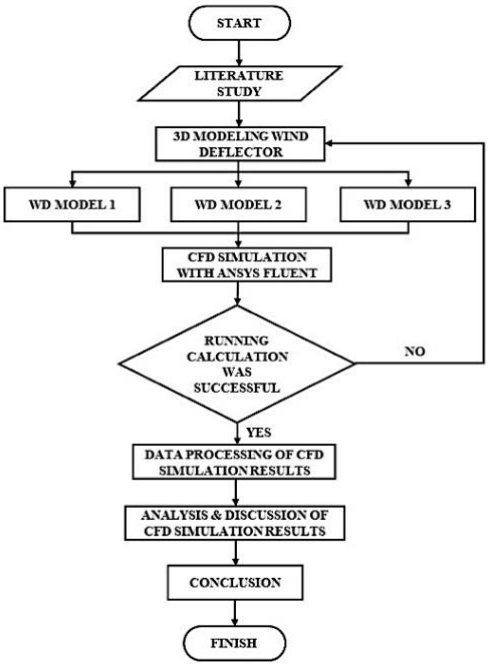


Figure 1. Research flowchart.

The research initiation involves an exhaustive exploration of the literature, entailing a comprehensive examination of diverse sources such as scholarly books and journals pertinent to the research context. Following this preliminary phase, the subsequent step entails the three-dimensional modeling process of Wind Deflector variations. This is executed through the utilization of computer-aided design (CAD)-based software. The modeling phase encompasses the meticulous design of three distinct model variations crafted to optimize Wind Deflector configurations specifically tailored for cargo van.

Consecutively, the subsequent stage involves the execution of CFD simulations on the three design variations of the Wind Deflector using Ansys Fluent R2 2023 software. These simulations are conducted under the assumption of a steady airflow velocity of 60 km/h. The results derived from the CFD simulations undergo automatic validation by the Ansys Fluent computational program for all three design variations. The validation of the simulation results is contingent upon the absence of error notifications or warnings, ensuring the reliability and accuracy of the computational outcomes.

Upon the successful completion of the CFD simulations meeting predefined criteria without any error or warning notifications, the ensuing step involves the processing and presentation of the generated data in tabular form. This systematic presentation is devised to facilitate a comprehensive comparative examination of the CFD simulation data across the three distinct design variations of the wind deflector. The comparative analysis stands as the cornerstone for the subsequent discussion and elaboration of the simulation data. This analytical phase forms the basis upon which the research conclusions are articulated. The research conclusions, in turn, encapsulate synthesized insights drawn from the comparative analysis, thereby contributing to the scholarly discourse on the optimization of wind deflector configurations for cargo van.

In the final segment of this research, the conclusions drawn will be accompanied by constructive recommendations aimed at guiding and informing potential avenues for further research endeavors in the field of aerodynamics

and vehicle design. These recommendations are intended to serve as valuable insights for researchers and practitioners seeking to advance the understanding and application of optimal wind deflector configurations for enhanced cargo van aerodynamics performance.

2.2. CFD Simulation Stages

The commencement of the CFD simulation process encompasses the initial stage of drawing a 3D model modification of the cargo van, incorporating wind deflector devices. This modification is illustrated in Figure 2. The Wind Deflector variations are realized by modifying the front box head angle to assume shapes resembling the Original Model, Model 1, Model 2, and Model 3, as depicted in Figure 2.

The subsequent stage involves the meshing process, wherein the solid geometry of each Wind Deflector device undergoes decomposition into smaller elements and nodes. Notably, the student version of ANSYS Fluent software imposes a limit of 1,000,000 cells/nodes in the meshing process (ANSYS, 2023). Hence, strategic determination of optimal meshing techniques becomes paramount at this juncture to ensure efficiency in the number of elements and nodes for all three variations of the Wind Deflector device models. The comprehensive breakdown of the elements and nodes resulting from the meshing process for each design variation is meticulously documented in Table 1.

Table 1 provides a comprehensive assessment of the efficiency regarding the number of elements and nodes within the three Wind Deflector model variations and the overall fluid domain. It is essential to underscore the consistent adoption of 25 mm dimensions for 3D tetrahedral elements (and 2D triangles) across all Wind Deflector model variations. This decision aligns with the elements' capabilities, strategically applied to remain within the prescribed maximum limit of 1,000,000 elements and nodes.

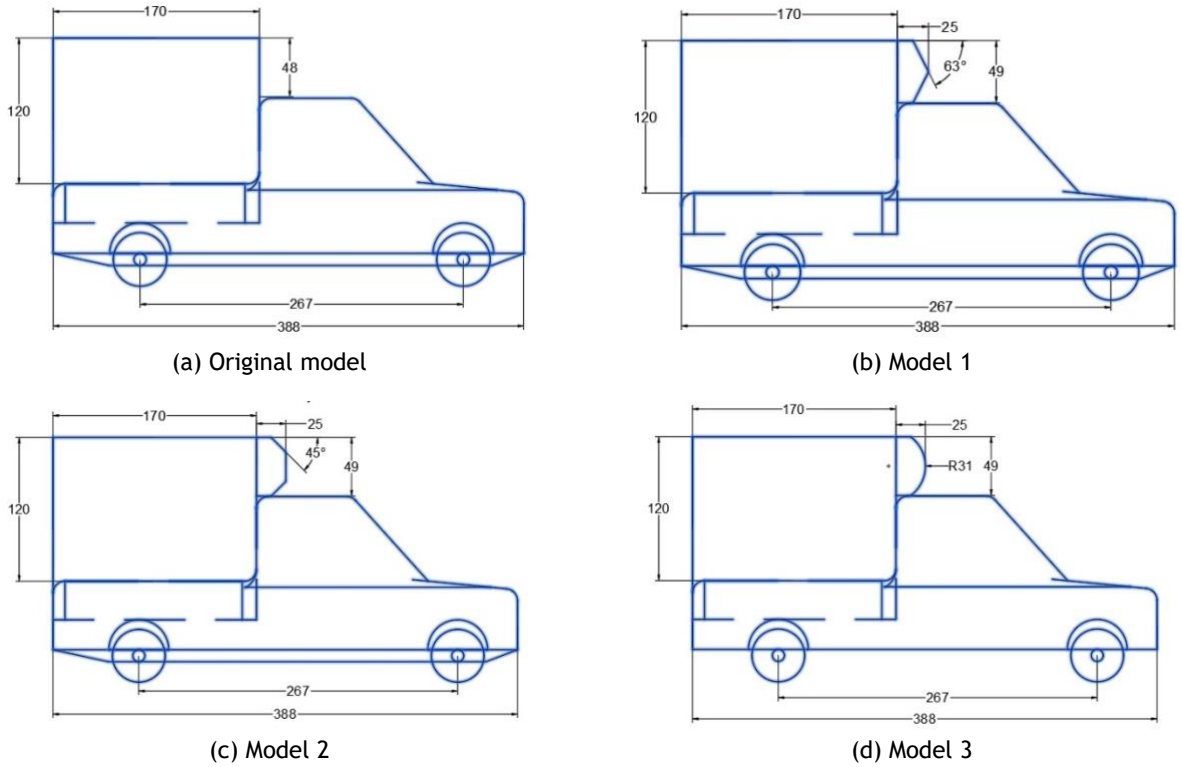


Figure 2. 3D Model modification of the cargo vans with wind deflector.

Table 1. Elements and nodes data.

Variant	Elements	Nodes
Original	982184	188781
Model 1	974100	187637
Model 2	974981	187781
Model 3	984326	189495

The mesh verification process assumes critical significance, ensuring that the utilization of a 25 mm element size yields an optimal and efficient meshing method. This verification entails a comparative analysis of the aspect ratio quality among different element sizes 25 mm, 35 mm, and 45 mm on a Model 1 test sample. The aspect ratio, quantifying the ratio between the longest and shortest lengths of each geometric edge, ideally approaches a value of 1 for optimal outcomes. Conversely, values approaching 20 denote poor aspects, particularly noticeable in triangular geometries, as illustrated in Figure 3 (Alawadhi, 2015).

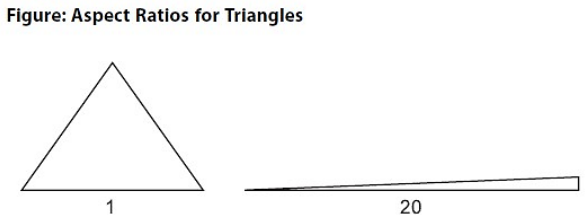


Figure 3. Ascpect ratio for triangles (Alawadhi, 2015).

Therefore, through a comparative analysis of standard deviation values and the smallest average aspect ratio across the three variations of element sizes, as illustrated in Table 2, insights into mesh quality are derived.

Table 2. Quality mesh data.

Element Size	Avarage Aspect Ratio	Standart Deviation
25 mm	1.9253	0.54886
35 mm	1.9291	0.55
45 mm	1.9626	0.59887

The data in Table 2 emanates from the outcomes of a mesh quality check conducted using the Ansys Meshing program. Given the limitations imposed by the maximum allowable mesh quantity in the Ansys Student version, the application of the 25 mm element size method emerges as both appropriate and efficient for deployment in the CFD simulation process. This determination is substantiated by the standard deviation values, where the smallest aspect ratio registers at 0.54886, and the smallest average aspect ratio attains a value of 1.9253. The mesh verification aligns seamlessly with the guidelines outlined in the Ansys Meshing User Guide, asserting that the optimal mesh exhibits decreasing aspect ratios approaching 1, while conversely, mesh quality deteriorates as aspect ratios increase towards 20 (Alawadhi, 2015).

The delineation of boundary conditions in CFD simulations involves the integration of parameter data gathered from diverse and credible literature sources. The comprehensive set of boundary conditions encompasses various crucial facets (Banga and Zunaid, 2017):

1) Viscous models: k-epsilon realizable non-equilibrium wall functions
2) Inlet velocity: 80 km/h
3) Initial gauge pressure: 0 Pa
4) Turbulent intensity: 5%
5) Turbulent viscosity ratio: 10
6) Air density: 1.225 kg/m^3
7) Air viscosity: 0.000017894 kg/ms

Additionally, certain parameters, such as solution methods and solution controls, are either maintained at default settings or regulated through the Ansys Fluent solver program. The simulation initialization process employs the hybrid initialization method. Subsequently, the run calculation process is executed with a parameter specifying the number of iterations set at 1000, ensuring the attainment of a maximally optimized calculation outcome.

Following the simulation, the results are visualized through the CFD-Post application. The results showcase the airflow traversing the cargo van body with Wind Deflector variations, visualized through distinctive color gradient contours and streamlines. Moreover, calculations provide values for the drag coefficient (Cd), drag force (Fd), pressure data, and wind pressure data experienced on each surface of the cargo van body with varying wind deflector configurations.

3. RESULTS AND DISCUSSION

The outcomes of the CFD simulation conducted with the ANSYS Fluent software are

presented in Table 3, illustrating the impact of Wind Deflector installation on drag force (N) and drag coefficient in a cargo van.

Table 3. Fd and Cd data of CFD simulation.

Variasi Desain	Drag Force (N)	Drag Coefficient
Original	1.6524553	0.53067397
Model 1	1.5896567	0.51038630
Model 2	1.2778974	0.41037838
Model 3	1.1531116	0.37031338

Analyzing the data in Table 3, it is discerned that the lowest drag force is recorded under the condition of the cargo van with the Wind Deflector Model 3 variation, amounting to 1.1531116 N. Conversely, the highest drag force is observed in the cargo van without a wind deflector (original) condition, measuring 1.6524553 N. Furthermore, the smallest drag coefficient value is identified in the cargo van condition with Wind Deflector Model 3 variation, registering at 0.37031338, while the highest drag coefficient is recorded in the cargo van without a wind deflector (original) condition, amounting to 0.53067397.

The visualization of CFD simulation results is presented in Figure 4 and Figure 5, illustrating the comparison of Wind Deflector (WD) variations concerning drag force (Fd) and drag coefficient (Cd).

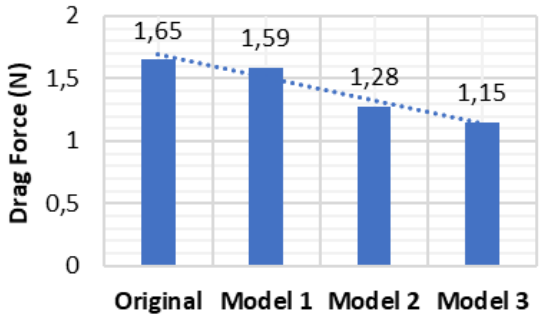


Figure 4. Drag force on cargo van with wind deflector variations.

Observing the data of drag force from the CFD simulation evaluation in Figure 4, it is evident that the application of Wind Deflector devices progressively reduces the drag force experienced by the cargo van. Wind Deflector modifies the pressure distribution on the vehicle surface, diminishes turbulent regions, and ultimately results in a reduction in drag force. The results of CFD simulation in this study indicate a decrease in drag force from the original model to the Wind Deflector Model 1 modification by 3.64%. Subsequently, in the Model 2 modification, there is a reduction of 22.24%, and in the final Model 3 modification, the reduction is 30.30%. These findings align with relevant prior research, which reported a 6.85% decrease in drag force with the installation of a wind deflector featuring a 5o angle variation on a cargo truck (Khosravi *et al.*, 2015). In essence, this study's results demonstrate consistency with earlier research regarding the reduction of drag force on cargo vehicles equipped with wind deflector devices. The magnitude of drag force reduction achieved is contingent upon the design or form of the wind deflector itself (Khosravi *et al.*, 2015).

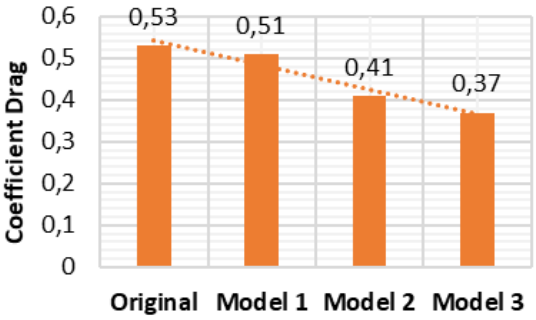


Figure 5. Drag coefficient on cargo van with wind deflector variations.

The data of drag coefficient from CFD simulation evaluation in Figure 5 indicates that variations in Wind Deflector significantly influence the aerodynamic characteristics of the cargo van. Changes in the shape and orientation of the Wind Deflector led to the redistribution of pressure, minimizing high-pressure zones that typically contribute to the drag coefficient. This phenomenon can be interpreted as a result of improved structural efficiency in mitigating aerodynamic resistance.

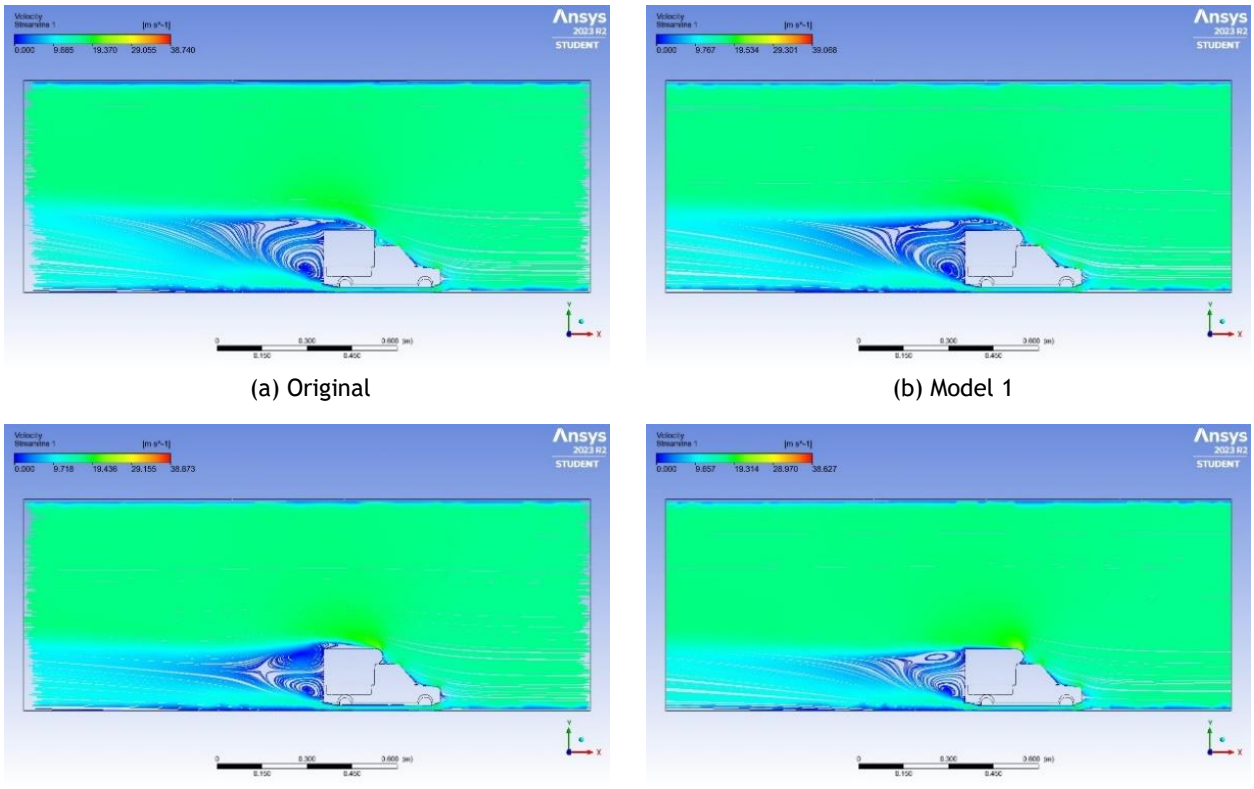
Therefore, the application of Wind Deflector devices with Model 3 variation enhances the aerodynamic design of the cargo van compared to Model 1, Model 2, and especially without Wind Deflector. Thus, the implementation of Wind Deflector on the cargo van can optimize aerodynamic performance by reducing drag force and drag coefficient.

The results of CFD simulation elucidate a decrease in the drag coefficient values for the cargo van equipped with Wind Deflector Model 1 by 3.77%, Model 2 by 22.67%, and Model 3 by 30.18%. These outcomes align with pertinent findings from previous research, which indicated a drag coefficient value of 0.651 for cargo trucks without wind deflectors. Subsequently, for cargo trucks with wind deflectors, the drag coefficient value was reduced to 0.542, resulting in a successful reduction of the drag coefficient by 16.74% (Hariram *et al.*, 2021).

The Computational Fluid Dynamics simulation results from ANSYS Fluent software also visualize the phenomenon of airflow through the cargo van body with each Wind Deflector variation, depicted by color gradient contouring as shown in Figure 6.

Contours that approach the red color indicate an increase in airflow velocity, while contours approaching the blue color signify a decrease in airflow velocity.

Observing Figure 6, it is evident that for each Wind Deflector design variation, the airflow velocity, as depicted in Figure 6, shows an increase in airflow around the Wind Deflector area, indicated by contours transitioning from green to approaching yellow. Additionally, there is a deceleration in airflow velocity at the front and rear sections of the cargo van body, illustrated by contours approaching dark blue. Furthermore, from the visualization of airflow rate for each Wind Deflector design variation in Figure 6, it is observed that there is a wake phenomenon from the upper to the rear part of the cargo van body.



(a) Original (b) Model 1

(c) Model 2 (d) Model 3

Figure 6. Visualization of airflow rate through the cargo van body with wind deflector.

The wake behind the vehicle tends to rotate towards the rear of the cargo vans and induces drag force on the vehicle, causing resistance to the vehicle's movement. The pressure difference at the front and rear of the cargo van body results in a deceleration of airflow. The faster the airflow separation occurs, the wider the wake becomes, leading to an increased drag force. Hence, it can

be elucidated that the cargo van equipped with the Wind Deflector Model 3 demonstrates a more aerodynamic design compared to other models. This is substantiated by the clear observation of a smaller wake area behind the cargo van with Wind Deflector Model 3, as depicted in Figure 6.

The findings of this research indicate that the cargo van equipped with Wind Deflector Model 3 has the smallest coefficient of drag (Cd) value, specifically at 0.37, with a 30.30% reduction in drag force compared to the cargo van without a wind deflector. In comparison to previous research, which reported the smallest Cd value at 0.508 and a 32.2% reduction in drag force for cargo trucks without wind deflectors, the current study shows promising results (Varadarajoo *et al.*, 2022).

The decrease in the Cd value for the cargo van can be attributed to the possibility of a more effective wind deflector design in directing airflow and minimizing aerodynamic resistance. Additionally, the box-like profile of the cargo van, which is less prominent than that of cargo trucks with a more dominant box-shaped profile in the front, contributes further to the reduction in Cd value and drag force. Therefore, it is recommended to explore the development of wind deflector designs that are more aerodynamic, capable of directing airflow to minimize turbulent flow effectively.

This research matchs with prior research conducted by Abdel-Salam and Chen, who performed a numerical analysis of airflow around a cargo van model using CFD (Abdel-Salam and Chen, 2014). Their findings revealed the presence of a significant wake region behind the van model, while this current study also highlights changes in drag force on a box vehicle with variations in Wind Deflector design. By comparing the findings of both studies, it can be explained that the additional design element, namely the Wind Deflector, has an impact on enhancing the aerodynamic performance of the box vehicle.

Furthermore, the findings of this study are connected to relevant previous research by Xu and Zhou, focusing on the analysis of airflow around trucks and the development of additional drag reduction devices to optimize aerodynamic performance (Xu and Zhou, 2018). The discovery that the addition of Wind Deflector devices results in a significant reduction in drag force aligns with earlier research findings that indicate drag reduction devices on trucks have a positive effect on reducing aerodynamic resistance.

Hence, this study significantly contributes to understanding the role of the Wind Deflector in the aerodynamic performance of cargo vans. Observations and analyses reveal that variations in Wind Deflector designs directly influence the airflow around the cargo van body, with Wind Deflector Model 3 emerging as a more aerodynamic choice. The installation of the Wind Deflector device can be considered an effective solution for reducing aerodynamic drag on cargo vans.

4. CONCLUSION

In summary, the comprehensive analysis of CFD simulation results in this study reveals a substantial enhancement in the aerodynamic performance of cargo vans with the installation of Wind Deflectors. Notably, Wind Deflector Model 3 stands out as the most effective design, showcasing a significant reduction of 30.30% in drag force and 30.18% in drag coefficient compared to the original configuration. The systematic examination of drag force, drag coefficient, and airflow patterns underscores the valuable contribution of Wind Deflectors in minimizing aerodynamic resistance and improving the overall efficiency of cargo vans.

Looking ahead, it is recommended to pursue further research and development in the realm of aerodynamic enhancements for cargo vehicles. Exploring additional Wind Deflector designs with optimized shapes and orientations could lead to even more significant improvements. The success of Wind Deflector Model 3 suggests the feasibility of broader applications across diverse vehicle types. To validate and refine the simulation results, physical wind tunnel testing is advisable, providing empirical data for a more robust understanding of the practical implications of Wind Deflector installations.

ACKNOWLEDGMENTS

The author on this occasion would like to thank the Kemendikbudristek for providing research funding assistance through the BIMA GRANT for the 2023 fiscal year with contract number 312/F.03.07/2023.

REFERENCES

Abdel-Salam, T. and Chen, D. (2014) 'A Computational Study of Cargo Van Aerodynamics', in *International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics*. *the 10th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics*, Orlando, Florida: International Centre for Heat and Mass (ICHMT), pp. 2239–2244.

Alawadhi, E.M. (2015) 'Meshing guide', in *Finite Element Simulations Using ANSYS*. 2nd edn. CRC Press. [Print].

ANSYS (2023) 'Ansys Student Software'. Available at: https://www.ansys.com/academic/students/ansys-student (Accessed: 29 December 2024).

Ariyansah, R. (2019) 'Modifikasi Desain Chasis Kendaraan Hybrid Pada Bus Scania K360LB', *Teknobiz : Jurnal Ilmiah Program Studi Magister Teknik Mesin*, 7(3), pp. 156–165.

Banga, S. and Zunaid, Md. (2017) 'CFD Simulation of Flow around External Vehicle: Ahmed Body', *IOSR Journal of Mechanical and Civil Engineering*, 12(04), pp. 87–94.

Gamayel, A. and Octavianus, G. (2021) *Tutorial Ansys Workbench untuk Bidang Mekanikal*. 2nd edn. Bandung, Indonesia: Media Sains Indonesia. [Print].

Hariram, A. *et al*. (2021) 'A Study in Options to Improve Aerodynamic Profile of Heavy-Duty Vehicles in Europe', *Sustainability*, 11(19), p. 5519.

Khosravi, M. *et al*. (2015) 'Aerodynamic Drag Reduction of Heavy Vehicles Using Append Devices by CFD Analysis', *Journal of Central South University*, 22(12), pp. 4645–4652.

Kshirsagar, V. and Chopade, J.V. (2018) 'Aerodynamics of High Performance Vehicles', *International Research Journal of Engineering and Technology (IRJET)*, 05(03), pp. 2182–2186.

Li, Y. *et al*. (2023) 'A Review on Numerical Simulation Based on CFD Technology of Aerodynamic Characteristics of Straight-Bladed Vertical Axis Wind Turbines', *Energy Reports*, 9, pp. 4360–4379.

McTavish, S. and McAuliffe, B. (2021) 'Improved Aerodynamic Fuel Savings Predictions for Heavy-Duty Vehicles Using Route-Specific Wind Simulations', *Journal of Wind Engineering and Industrial Aerodynamics*, 210, p. 104528.

Moriaand, H. and Alam, F. (2012) 'A Computational Simulation of Aerodynamic Drag Reductions for Heavy Commercial Vehicles', in *18th Australasian Fluid Mechanics Conference*. *18th Australasian Fluid Mechanics Conference*, Launceston, Australia, pp. 1–4.

Nath, D.S. *et al*. (2021) 'Drag Reduction by Application of Aerodynamic Devices in A Race Car', *Advances in Aerodynamics*, 3(1), p. 4.

Paminto, A.K. (2020) 'Analisis Dan Proyeksi Kebutuhan Energi Sektor Transportasi Di Indonesia', *Jurnal Energi dan Lingkungan*, 16(2), pp. 51–54.

Tu, J. *et al*. (2023) *Computational Fluid Dynamics: A Practical Approach*. India: Elsevier. [Print].

Varadarajoo, P.K. *et al*. (2022) 'Aerodynamic Analysis on the Effects of Frontal Deflector on a Truck by using Ansys Software', *International Journal of Integrated Engineering*, 14(6), pp. 77–87.

Xu, J. and Zhou, S. (2018) 'Flow Field Analysis of Trucks and a Design of an Addtional Drag Reduction Device', *Engineering Review*, pp. 70–78.

Yulianto, F.A., Ariyansah, R. and Octavianus, G. (2023) 'Analisis Modifikasi Desain pada Lokomotif CC-201 Tipe GE U18C dengan Metode Simulasi CFD', *Jurnal Asiimetrik: Jurnal Ilmiah Rekayasa dan Inovasi*, 5(2), pp. 161–170.

Layanan Perpustakaan UHAMKA

Evaluating Wind Deflector Effect on Cargo Vans Aerodynamic Drag Using Computational Fluid Dynamics.pdf

LK BU OKTA
Fakultas Teknologi Industri dan Informatika
Universitas Muhammadiyah Prof. Dr. Hamka

Document Details

Submission ID
trn:oid:::1:3239884463

Submission Date
May 5, 2025, 4:35 PM GMT+7

Download Date
May 15, 2025, 10:48 AM GMT+7

File Name
Evaluating_Wind_Deflector_Effect_on_Cargo_Vans_Aerodynamic_Drag_Using_Computational_Flui....pdf

File Size
762.2 KB

10 Pages

5,216 Words

30,050 Characters

5% Overall Similarity

The combined total of all matches, including overlapping sources, for each database.

Filtered from the Report

- Bibliography

Exclusions

- 1 Excluded Source
- 20 Excluded Matches

Match Groups

21 Not Cited or Quoted 4%
Matches with neither in-text citation nor quotation marks

6 Missing Quotations 1%
Matches that are still very similar to source material

0 Missing Citation 0%
Matches that have quotation marks, but no in-text citation

0 Cited and Quoted 0%
Matches with in-text citation present, but no quotation marks

Top Sources

3% Internet sources
3% Publications
0% Submitted works (Student Papers)

Integrity Flags

0 Integrity Flags for Review

No suspicious text manipulations found.

Our system's algorithms look deeply at a document for any inconsistencies that would set it apart from a normal submission. If we notice something strange, we flag it for you to review.

A Flag is not necessarily an indicator of a problem. However, we'd recommend you focus your attention there for further review.

Match Groups

- **21** Not Cited or Quoted 4%
  Matches with neither in-text citation nor quotation marks
- **6** Missing Quotations 1%
  Matches that are still very similar to source material
- **0** Missing Citation 0%
  Matches that have quotation marks, but no in-text citation
- **0** Cited and Quoted 0%
  Matches with in-text citation present, but no quotation marks

Top Sources

3% Internet sources
3% Publications
0% Submitted works (Student Papers)

Top Sources

The sources with the highest number of matches within the submission. Overlapping sources will not be displayed.

1	Publication	Sean McTavish, Brian McAuliffe. "Improved aerodynamic fuel savings predictions ... <1%
2	Internet	www.researchgate.net <1%
3	Internet	www.mdpi.com <1%
4	Publication	"Scientific and Technological Advances in Materials for Energy Storage and Conve... <1%
5	Internet	openaccess.thecvf.com <1%
6	Internet	eprints.utm.my <1%
7	Internet	journals.lww.com <1%
8	Internet	kar.kent.ac.uk <1%
9	Publication	Pranav Anil Hol, Mohammad Rafiq Agrewale. "Aerodynamic Analysis of Passenge... <1%
10	Internet	repository.uhamka.ac.id <1%

11 Internet

tasarimkuram.com <1%

12 Publication

Johannes Haff, Mattias Jönsson, Sigfried Loose, Claus Wagner. "Reliability of Engi... <1%

13 Publication

Katrina Calautit, Cameron Johnstone. "Investigation of the integration of an oscill... <1%

14 Internet

discovery.researcher.life <1%

15 Publication

"The 30th SIAR International Congress of Automotive and Transport Engineering"... <1%

16 Publication

Iacopo Carnacina, Mawada Abdellatif, Manolia Andredaki, James Cooper, Darren ... <1%

17 Publication

Sean McTavish, Fa Ghorbanishohrat, Brian McAuliffe. "Measurement of the on-ro... <1%

18 Publication

Stephen F. McCormick. "Multigrid Methods - Theory, Applications, and Supercom... <1%

19 Publication

Yasser Mahmoudi, Kamel Hooman, Kambiz Vafai. "Convective Heat Transfer in Po... <1%

20 Internet

dspace.vutbr.cz <1%

21 Internet

mdpi-res.com <1%

22 Internet

pt.scribd.com <1%

Jurnal Asiimetrik: Jurnal Ilmiah Rekayasa Dan Inovasi
Terakreditasi "**Peringkat 3**" berdasarkan SK Dirjendiktiristek, Nomor: 225/E/KPT/2022
Volume 6 Nomor 2 Tahun 2024, Juli 2024, Halaman 211-220
Website : http://journal.univpancasila.ac.id/index.php/asiimetrik/
p-ISSN 2655-1861
e-ISSN 2716-2923

Evaluating Wind Deflector Effect on Cargo Vans Aerodynamic Drag Using Computational Fluid Dynamics

Evaluasi Pengaruh *Wind Deflector* terhadap Gaya Hambat Aerodinamis Mobil Box dengan Dinamika Fluida Komputasional

Agus Fikri[1], Riyan Ariyansah[1]*, Firman Noor Hasan[2], Oktarina Heriyani[1], Rosalina[3], Muhammad Ghiffar Sistani[1]
[1]Program Studi Teknik Mesin, Fakultas Teknologi Industri dan Informatika, Universitas Muhammadiyah Prof. DR. HAMKA, Jl. Tanah Merdeka, No.6, Ciracas, Jakarta Timur, Indonesia
[2]Program Studi Teknik Informatika, Fakultas Teknologi Industri dan Informatika, Universitas Muhammadiyah Prof. DR. HAMKA, Jl. Tanah Merdeka, No.6, Ciracas, Jakarta Timur, Indonesia
[3]Program Studi Teknik Elektro, Fakultas Teknologi Industri dan Informatika, Universitas Muhammadiyah Prof. DR. HAMKA, Jl. Tanah Merdeka, No.6, Ciracas, Jakarta Timur, Indonesia

Article information:

Received:
04/01/2024
Revised:
04/03/2024
Accepted:
19/03/2024

Abstract

Suboptimal design and body shape in freight transport vehicles can lead to increased aerodynamic drag. To address this issue, the use of wind deflectors is proposed as a solution to reduce aerodynamic resistance in cargo vans. The methodology employed in this research involves Computational Fluid Dynamics (CFD) simulations using the Ansys Fluent R2 2023 software. CFD simulations were conducted on the design of a cargo box vehicle with variations in Wind Deflector Models 1, 2, and 3, employing identical boundary condition parameters. The results of the CFD simulation for Wind Deflector Model 3 exhibited the lowest drag force at 1.1531116 Newton and a drag coefficient of 0.37031338. In conclusion, a comprehensive analysis of the CFD simulation results provides valuable insights into the intricate aerodynamic implications of Wind Deflector variations on cargo vans. Therefore, it is concluded that Wind Deflector Model 3 emerges as the optimal choice, showcasing superior aerodynamic characteristics.

Keywords: cargo van, wind deflector, CFD simulation, drag force, drag coefficient.

SDGs:

Abstrak

Desain dan bentuk bodi yang tidak optimal pada kendaraan angkutan barang dapat menyebabkan peningkatan hambatan udara. Untuk mengatasi masalah ini, penggunaan *wind deflector* diusulkan sebagai solusi untuk mengurangi hambatan aerodinamis pada van kargo. Metodologi yang digunakan dalam penelitian ini melibatkan simulasi CFD menggunakan perangkat lunak Ansys Fluent R2 2023. Simulasi CFD dilakukan pada desain mobil box kargo dengan variasi bentuk *Wind Deflector* Model 1, Model 2, dan Model 3, dengan menggunakan parameter kondisi batas yang identik. Hasil simulasi CFD *Wind Deflector* Model 3 menunjukkan gaya hambat terendah sebesar 1.1531116 Newton dan koefisien hambat sebesar 0.37031338. Sebagai kesimpulan, analisis menyeluruh terhadap hasil simulasi CFD memberikan wawasan berharga tentang implikasi aerodinamis yang rumit dari variasi *Wind Deflector* pada mobil van kargo, maka didapat kesimpulan bahwa *Wind Deflector* Model 3 muncul sebagai pilihan optimal menampilkan karakteristik aerodinamis yang unggul.

Kata Kunci: mobil box, *wind deflector*, simulasi CFD, gaya hambat, koefiesien hambat.

*Correspondence Author
email : riyan_ariyansah@uhamka.ac.id

This work is licensed under a Creative Commons Attribution-NonCommercial 4.0 International License

1. INTRODUCTION

In the era of modern transportation, mobility emerges as a crucial element supporting economic growth and social comfort. The utilization of motorized vehicles, particularly in the realm of freight transportation, has surged alongside the increasing trends in trade and distribution (Paminto, 2020). Consequently, technologies related to freight transport vehicles have also undergone significant advancements. However, the design of freight transport vehicles, including cargo vans characterized by their predominantly box-like shape, remains a focal point in efforts to enhance aerodynamic efficiency. In this context, the implementation of wind deflector devices on cargo vans becomes a highly relevant factor in reducing the coefficient of drag and overall aerodynamic resistance. This, in turn, contributes to the development of cargo van designs that are more aerodynamic.

To comprehend the inherent aerodynamic challenges faced by cargo van, an essential grasp of the theory concerning drag force becomes imperative. The aerodynamic drag force acting on a vehicle is the resultant effect of air resistance encountered during the vehicle's movement through the atmospheric medium (Nath *et al.*, 2021). The deployment of Wind Deflectors emerges as a pragmatic strategy to mitigate aerodynamic drag in vehicles (Hariram *et al.*, 2021). These devices, modifiable or installable at specific sections of the vehicle, are meticulously crafted to redirect airflow, thereby ameliorating drag force comprehensively. The substantial reduction of drag force on cargo vans assumes pivotal importance in enhancing the sustainability of cargo transportation. Cargo vans, including trucks and freight-carrying vehicles, attract attention owing to their inherent aerodynamic profiles, which tend to instigate higher drag forces. The utilization of Wind Deflectors, also known as air shields, has become a prevalent strategy in endeavors aimed at refining the aerodynamic efficiency of these vehicles.

The application of CFD simulations confers a strategic advantage by enabling the virtual testing of diverse Wind Deflector configurations and other design variations on cargo vans. This capability permits the exploration of potential aerodynamic strategies without the need for resource intensive physical prototype production (Ariyansah, 2019). CFD represents a numerical approach widely employed for modeling and analyzing fluid flow around objects with intricate geometries, obviating the need for costly and intricate physical testing (Gamayel and Octavianus, 2021).

The underlying theory of CFD finds its roots in the Navier-Stokes equations a set of partial differential equations formulated to model fluid behavior (Li *et al.*, 2023). Employing discretization methods such as Finite Volume or Finite Element, the numerical solution of the Navier-Stokes equations yields comprehensive insights into pressure distribution, velocity, and other aerodynamic parameters along the surface of the vehicle (Kshirsagar and Chopade, 2018). Thus, CFD serves as a powerful tool for accurate simulations elucidating the intricate interaction between airflow and cargo vans.

The ANSYS Fluent software is widely utilized for CFD simulations in various research scenarios, particularly for analyzing the aerodynamics of vehicles. A study conducted by Yulianto, Ariyansah, and Octavianus in 2023 employed the ANSYS Fluent software version R2 2022 to investigate the influence of design angle variations on the U18C CC-201 locomotive's drag coefficient and aerodynamics (Yulianto, Ariyansah and Octavianus, 2023). The research findings elucidate that a 150-degree angle modification on the upper part of the GE U18C locomotive body has a significant impact, resulting in a notable reduction of up to 29% in the drag coefficient.

Furthermore, the study conducted by Varadarajoo et al., delves into the aerodynamic analysis of the impact of the Frontal Deflector on trucks using the ANSYS Fluent software (Varadarajoo *et al.*, 2022). The drag force generated by airflow separation at the rear of the vehicle can be mitigated through the implementation of a Frontal Deflector. Despite creating drag, this research demonstrates that the Frontal Deflector can reduce drag by preventing airflow separation at the rear of the truck. Simulations involving six different forms of Frontal Deflector reveal that Frontal Deflector 4 exhibits

the lowest drag coefficient, reaching 0.508 with a height of 15 mm and a placement of 230 mm. With a drag reduction of 32.2%, Frontal Deflector 4 is recognized as the optimal model for enhancing the aerodynamic efficiency of trucks.

The pursuit of optimizing aerodynamic efficiency in cargo vans has garnered considerable attention in prior research, particularly with an emphasis on the effectiveness of Wind Deflectors. Pevitt et al., conducted a seminal study, showcasing the successful application of CFD simulations on heavy commercial vehicles (Moriaand and Alam, 2012). This work has laid a robust methodological foundation for the comprehensive analysis of aerodynamic characteristics. Notably, this methodology facilitates detailed visual modeling of airflow patterns around the vehicle and offers adaptability to various experimental conditions within the virtual realm.

The contribution of Moriaand and Alam, work serves as a noteworthy precedent, illustrating the efficacy of CFD simulations in gaining a nuanced understanding of aerodynamic features in commercial vehicles (Moriaand and Alam, 2012). By employing this methodology, researchers can conduct virtual experiments with a high degree of accuracy, exploring diverse scenarios and configurations to optimize the design of Wind Deflectors for enhanced aerodynamic performance in cargo vans. This approach not only advances scientific knowledge in aerodynamics but also presents a cost-effective and efficient means to refine and innovate vehicle design without the necessity for extensive physical prototypes.

McTavish and McAuliffe, not only emphasize the broader potential for aerodynamic optimization in heavy-duty vehicles but also provide a contextual backdrop for endeavors aimed at enhancing fuel efficiency (McTavish and McAuliffe, 2021). This study introduces a stochastic simulation methodology to refine the accurate estimation of the efficiency of aerodynamic drag reduction technologies in ground vehicles, particularly heavy-duty vehicles. This approach utilizes specific wind conditions on designated routes to predict the speed and yaw angle of the vehicle. Predictions of aerodynamic drag and fuel consumption are grounded in the vehicle's aerodynamic performance model derived from wind tunnel measurements, road measurements, or CFD simulations.

Simulations are employed to assess the benefits of drag reduction technologies in heavy-duty vehicles on two main routes in Canada, illustrating that fuel savings can be achieved at low speeds, such as 50 km/h, with a noteworthy contribution from tractor-trailer platooning approaches. Evaluation across various vehicle weights indicates that, even as the proportion of fuel consumption attributed to aerodynamics decreases at low speeds, drag reduction continues to yield substantial fuel savings. This not only supports fuel cost reduction but also contributes to the mitigation of greenhouse gas emissions.

Despite numerous studies investigating the implementation of Wind Deflectors on motor vehicles and their impact on aerodynamic resistance, there exists a notable research gap, particularly in the context of cargo vans. Therefore, there is an urgent need to fill this knowledge gap and delve into how Wind Deflectors can be optimized to enhance aerodynamic efficiency in cargo vans, which, in turn, can significantly contribute to the sustainability of freight transportation. The novelty of this research lies in its specific focus on cargo vans and the utilization of CFD technology as the primary tool to investigate the complex aerodynamic phenomena associated with these vehicles. This study aims to analyze the effects of Wind Deflectors on the aerodynamic drag force of cargo vans. By achieving the objectives of this research, it is anticipated to provide profound insights into how the addition of Wind Deflector devices can improve aerodynamic efficiency in cargo vans.

2. METHODOLOGY

In this study, the primary methodology employed is the CFD simulation approach. However, it is imperative to emphasize that the implemented CFD simulation approach is not utilized in isolation rather, it is intricately connected to both experimental and analytical methodologies.

This synergistic integration is strategically devised to address potential errors and deficiencies in parameter data during the execution of CFD simulations, with the overarching goal of mitigating, if not eliminating, uncertainties inherent in the research process (Tu *et al.*, 2023).

This strategic integration unfolds avenues for deriving comprehensive advantages from the amalgamation of empirically verifiable experimental data, in-depth mathematical analyses, and advanced numerical simulations. Consequently, this methodological framework establishes a robust foundation, thereby enhancing the reliability and credibility of the research outcomes. The amalgamation of these three approaches provides the requisite flexibility to delve into specific facets that may pose challenges if approached solely through one method. Therefore, the success of this research is contingent not solely upon the excellence of the CFD simulation approach but also on the harmonious and synergistic integration with experimental and analytical methodologies.

2.1. Research Flow

The progression of this investigation is elucidated in the research flowchart as shown in Figure 1.

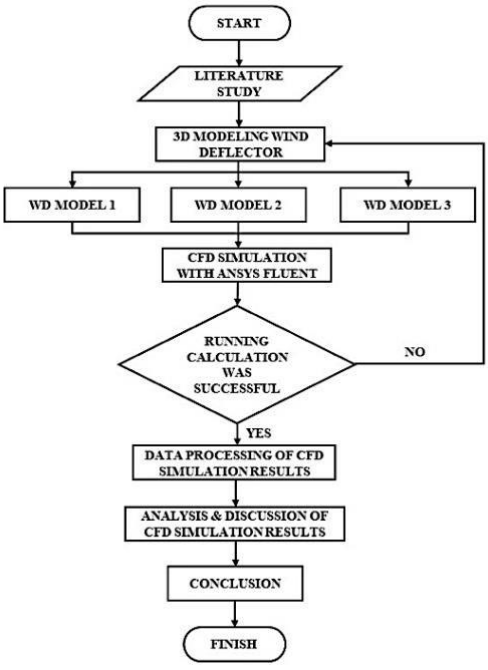


Figure 1. Research flowchart.

The research initiation involves an exhaustive exploration of the literature, entailing a comprehensive examination of diverse sources such as scholarly books and journals pertinent to the research context. Following this preliminary phase, the subsequent step entails the three-dimensional modeling process of Wind Deflector variations. This is executed through the utilization of computer-aided design (CAD)-based software. The modeling phase encompasses the meticulous design of three distinct model variations crafted to optimize Wind Deflector configurations specifically tailored for cargo van.

Consecutively, the subsequent stage involves the execution of CFD simulations on the three design variations of the Wind Deflector using Ansys Fluent R2 2023 software. These simulations are conducted under the assumption of a steady airflow velocity of 60 km/h. The results derived from the CFD simulations undergo automatic validation by the Ansys Fluent computational program for all three design variations. The validation of the simulation results is contingent upon the absence of error notifications or warnings, ensuring the reliability and accuracy of the computational outcomes.

Upon the successful completion of the CFD simulations meeting predefined criteria without any error or warning notifications, the ensuing step involves the processing and presentation of the generated data in tabular form. This systematic presentation is devised to facilitate a comprehensive comparative examination of the CFD simulation data across the three distinct design variations of the wind deflector. The comparative analysis stands as the cornerstone for the subsequent discussion and elaboration of the simulation data. This analytical phase forms the basis upon which the research conclusions are articulated. The research conclusions, in turn, encapsulate synthesized insights drawn from the comparative analysis, thereby contributing to the scholarly discourse on the optimization of wind deflector configurations for cargo van.

In the final segment of this research, the conclusions drawn will be accompanied by constructive recommendations aimed at guiding and informing potential avenues for further research endeavors in the field of aerodynamics

and vehicle design. These recommendations are intended to serve as valuable insights for researchers and practitioners seeking to advance the understanding and application of optimal wind deflector configurations for enhanced cargo van aerodynamics performance.

2.2. CFD Simulation Stages

The commencement of the CFD simulation process encompasses the initial stage of drawing a 3D model modification of the cargo van, incorporating wind deflector devices. This modification is illustrated in Figure 2. The Wind Deflector variations are realized by modifying the front box head angle to assume shapes resembling the Original Model, Model 1, Model 2, and Model 3, as depicted in Figure 2.

The subsequent stage involves the meshing process, wherein the solid geometry of each Wind Deflector device undergoes decomposition into smaller elements and nodes. Notably, the student version of ANSYS Fluent software imposes a limit of 1,000,000 cells/nodes in the meshing process (ANSYS, 2023). Hence, strategic determination of optimal meshing techniques becomes paramount at this juncture to ensure efficiency in the number of elements and nodes for all three variations of the Wind Deflector device models. The comprehensive breakdown of the elements and nodes resulting from the meshing process for each design variation is meticulously documented in Table 1.

Table 1 provides a comprehensive assessment of the efficiency regarding the number of elements and nodes within the three Wind Deflector model variations and the overall fluid domain. It is essential to underscore the consistent adoption of 25 mm dimensions for 3D tetrahedral elements (and 2D triangles) across all Wind Deflector model variations. This decision aligns with the elements' capabilities, strategically applied to remain within the prescribed maximum limit of 1,000,000 elements and nodes.

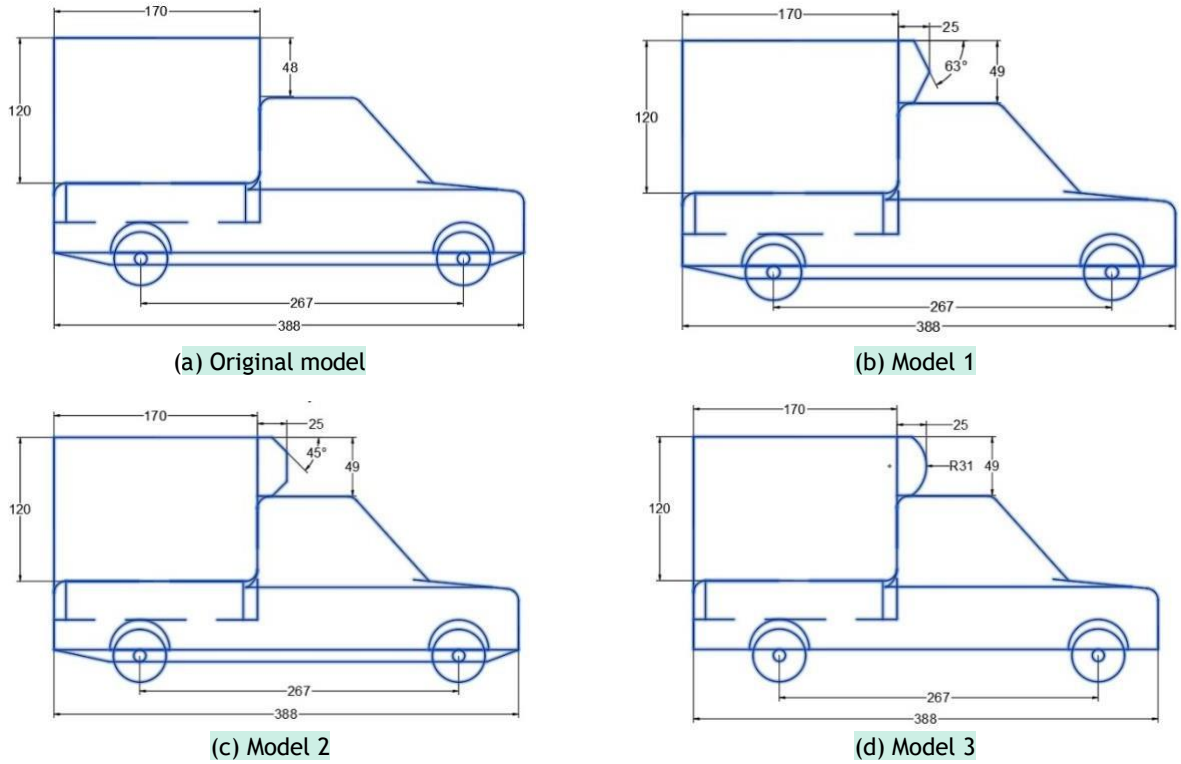


Figure 2. 3D Model modification of the cargo vans with wind deflector.

Table 1. Elements and nodes data.

Variant	Elements	Nodes
Original	982184	188781
Model 1	974100	187637
Model 2	974981	187781
Model 3	984326	189495

The mesh verification process assumes critical significance, ensuring that the utilization of a 25 mm element size yields an optimal and efficient meshing method. This verification entails a comparative analysis of the aspect ratio quality among different element sizes 25 mm, 35 mm, and 45 mm on a Model 1 test sample. The aspect ratio, quantifying the ratio between the longest and shortest lengths of each geometric edge, ideally approaches a value of 1 for optimal outcomes. Conversely, values approaching 20 denote poor aspects, particularly noticeable in triangular geometries, as illustrated in Figure 3 (Alawadhi, 2015).

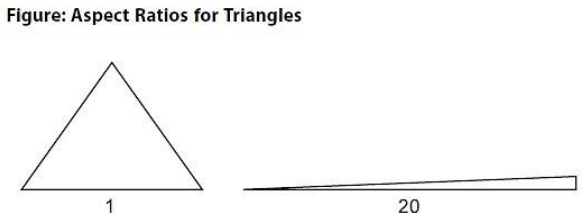


Figure 3. Ascpect ratio for triangles (Alawadhi, 2015).

Therefore, through a comparative analysis of standard deviation values and the smallest average aspect ratio across the three variations of element sizes, as illustrated in Table 2, insights into mesh quality are derived.

Table 2. Quality mesh data.

Element Size	Avarage Aspect Ratio	Standart Deviation
25 mm	1.9253	0.54886
35 mm	1.9291	0.55
45 mm	1.9626	0.59887

The data in Table 2 emanates from the outcomes of a mesh quality check conducted using the Ansys Meshing program. Given the limitations imposed by the maximum allowable mesh quantity in the Ansys Student version, the application of the 25 mm element size method emerges as both appropriate and efficient for deployment in the CFD simulation process. This determination is substantiated by the standard deviation values, where the smallest aspect ratio registers at 0.54886, and the smallest average aspect ratio attains a value of 1.9253. The mesh verification aligns seamlessly with the guidelines outlined in the Ansys Meshing User Guide, asserting that the optimal mesh exhibits decreasing aspect ratios approaching 1, while conversely, mesh quality deteriorates as aspect ratios increase towards 20 (Alawadhi, 2015).

The delineation of boundary conditions in CFD simulations involves the integration of parameter data gathered from diverse and credible literature sources. The comprehensive set of boundary conditions encompasses various crucial facets (Banga and Zunaid, 2017):

1) Viscous models: k-epsilon realizable non-equilibrium wall functions
2) Inlet velocity: 80 km/h
3) Initial gauge pressure: 0 Pa
4) Turbulent intensity: 5%
5) Turbulent viscosity ratio: 10
6) Air density: 1.225 kg/m^3
7) Air viscosity: 0.000017894 kg/ms

Additionally, certain parameters, such as solution methods and solution controls, are either maintained at default settings or regulated through the Ansys Fluent solver program. The simulation initialization process employs the hybrid initialization method. Subsequently, the run calculation process is executed with a parameter specifying the number of iterations set at 1000, ensuring the attainment of a maximally optimized calculation outcome.

Following the simulation, the results are visualized through the CFD-Post application. The results showcase the airflow traversing the cargo van body with Wind Deflector variations, visualized through distinctive color gradient contours and streamlines. Moreover, calculations provide values for the drag coefficient (Cd), drag force (Fd), pressure data, and wind pressure data experienced on each surface of the cargo van body with varying wind deflector configurations.

3. RESULTS AND DISCUSSION

The outcomes of the CFD simulation conducted with the ANSYS Fluent software are

presented in Table 3, illustrating the impact of Wind Deflector installation on drag force (N) and drag coefficient in a cargo van.

Table 3. Fd and Cd data of CFD simulation.

Variasi Desain	Drag Force (N)	Drag Coefficient
Original	1.6524553	0.53067397
Model 1	1.5896567	0.51038630
Model 2	1.2778974	0.41037838
Model 3	1.1531116	0.37031338

Analyzing the data in Table 3, it is discerned that the lowest drag force is recorded under the condition of the cargo van with the Wind Deflector Model 3 variation, amounting to 1.1531116 N. Conversely, the highest drag force is observed in the cargo van without a wind deflector (original) condition, measuring 1.6524553 N. Furthermore, the smallest drag coefficient value is identified in the cargo van condition with Wind Deflector Model 3 variation, registering at 0.37031338, while the highest drag coefficient is recorded in the cargo van without a wind deflector (original) condition, amounting to 0.53067397.

The visualization of CFD simulation results is presented in Figure 4 and Figure 5, illustrating the comparison of Wind Deflector (WD) variations concerning drag force (Fd) and drag coefficient (Cd).

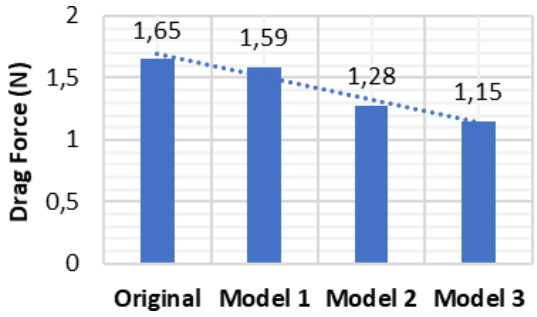


Figure 4. Drag force on cargo van with wind deflector variations.

Observing the data of drag force from the CFD simulation evaluation in Figure 4, it is evident that the application of Wind Deflector devices progressively reduces the drag force experienced by the cargo van. Wind Deflector modifies the pressure distribution on the vehicle surface, diminishes turbulent regions, and ultimately results in a reduction in drag force. The results of CFD simulation in this study indicate a decrease in drag force from the original model to the Wind Deflector Model 1 modification by 3.64%. Subsequently, in the Model 2 modification, there is a reduction of 22.24%, and in the final Model 3 modification, the reduction is 30.30%. These findings align with relevant prior research, which reported a 6.85% decrease in drag force with the installation of a wind deflector featuring a 5o angle variation on a cargo truck (Khosravi *et al.*, 2015). In essence, this study's results demonstrate consistency with earlier research regarding the reduction of drag force on cargo vehicles equipped with wind deflector devices. The magnitude of drag force reduction achieved is contingent upon the design or form of the wind deflector itself (Khosravi *et al.*, 2015).

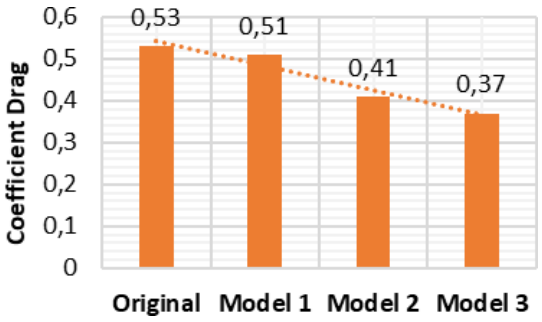


Figure 5. Drag coefficient on cargo van with wind deflector variations.

The data of drag coefficient from CFD simulation evaluation in Figure 5 indicates that variations in Wind Deflector significantly influence the aerodynamic characteristics of the cargo van. Changes in the shape and orientation of the Wind Deflector led to the redistribution of pressure, minimizing high-pressure zones that typically contribute to the drag coefficient. This phenomenon can be interpreted as a result of improved structural efficiency in mitigating aerodynamic resistance.

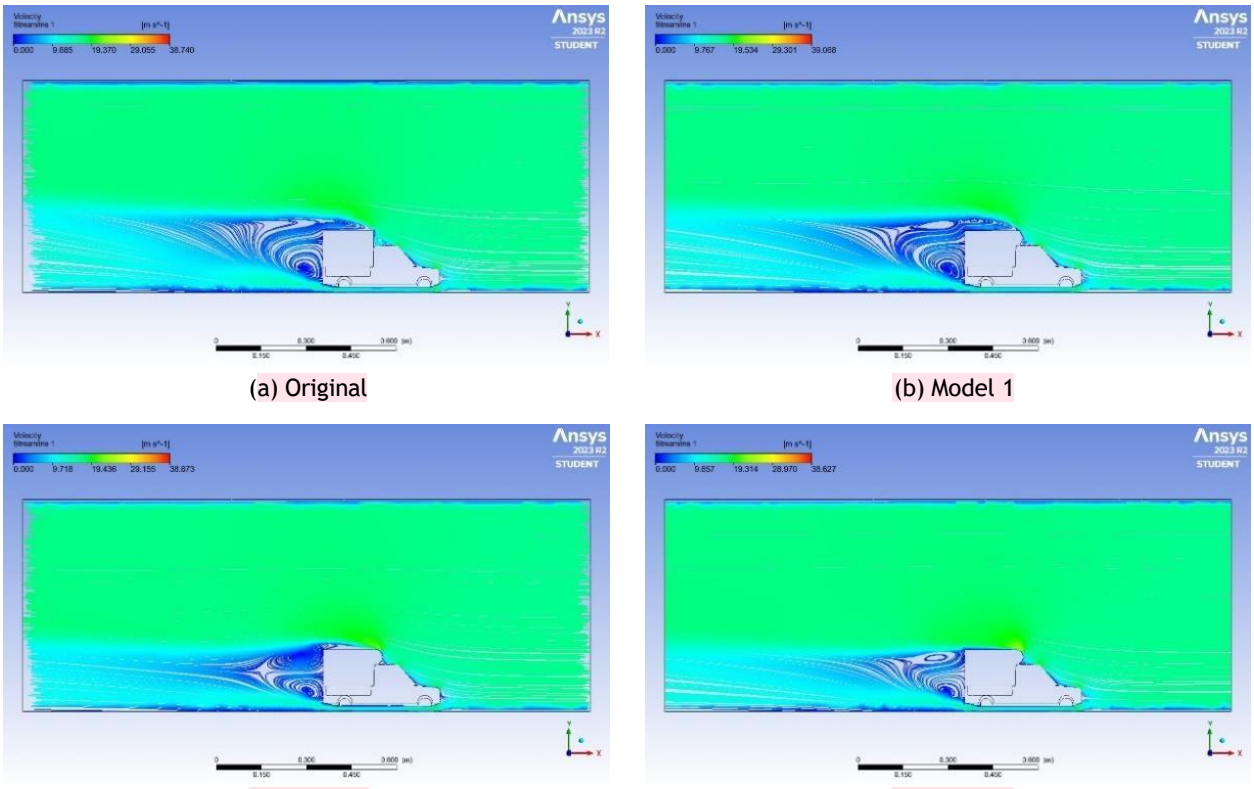
Therefore, the application of Wind Deflector devices with Model 3 variation enhances the aerodynamic design of the cargo van compared to Model 1, Model 2, and especially without Wind Deflector. Thus, the implementation of Wind Deflector on the cargo van can optimize aerodynamic performance by reducing drag force and drag coefficient.

The results of CFD simulation elucidate a decrease in the drag coefficient values for the cargo van equipped with Wind Deflector Model 1 by 3.77%, Model 2 by 22.67%, and Model 3 by 30.18%. These outcomes align with pertinent findings from previous research, which indicated a drag coefficient value of 0.651 for cargo trucks without wind deflectors. Subsequently, for cargo trucks with wind deflectors, the drag coefficient value was reduced to 0.542, resulting in a successful reduction of the drag coefficient by 16.74% (Hariram *et al.*, 2021).

The Computational Fluid Dynamics simulation results from ANSYS Fluent software also visualize the phenomenon of airflow through the cargo van body with each Wind Deflector variation, depicted by color gradient contouring as shown in Figure 6.

Contours that approach the red color indicate an increase in airflow velocity, while contours approaching the blue color signify a decrease in airflow velocity.

Observing Figure 6, it is evident that for each Wind Deflector design variation, the airflow velocity, as depicted in Figure 6, shows an increase in airflow around the Wind Deflector area, indicated by contours transitioning from green to approaching yellow. Additionally, there is a deceleration in airflow velocity at the front and rear sections of the cargo van body, illustrated by contours approaching dark blue. Furthermore, from the visualization of airflow rate for each Wind Deflector design variation in Figure 6, it is observed that there is a wake phenomenon from the upper to the rear part of the cargo van body.



(a) Original (b) Model 1

(c) Model 2 (d) Model 3

Figure 6. Visualization of airflow rate through the cargo van body with wind deflector.

The wake behind the vehicle tends to rotate towards the rear of the cargo vans and induces drag force on the vehicle, causing resistance to the vehicle's movement. The pressure difference at the front and rear of the cargo van body results in a deceleration of airflow. The faster the airflow separation occurs, the wider the wake becomes, leading to an increased drag force. Hence, it can

be elucidated that the cargo van equipped with the Wind Deflector Model 3 demonstrates a more aerodynamic design compared to other models. This is substantiated by the clear observation of a smaller wake area behind the cargo van with Wind Deflector Model 3, as depicted in Figure 6.

The findings of this research indicate that the cargo van equipped with Wind Deflector Model 3 has the smallest coefficient of drag (Cd) value, specifically at 0.37, with a 30.30% reduction in drag force compared to the cargo van without a wind deflector. In comparison to previous research, which reported the smallest Cd value at 0.508 and a 32.2% reduction in drag force for cargo trucks without wind deflectors, the current study shows promising results (Varadarajoo *et al.*, 2022).

The decrease in the Cd value for the cargo van can be attributed to the possibility of a more effective wind deflector design in directing airflow and minimizing aerodynamic resistance. Additionally, the box-like profile of the cargo van, which is less prominent than that of cargo trucks with a more dominant box-shaped profile in the front, contributes further to the reduction in Cd value and drag force. Therefore, it is recommended to explore the development of wind deflector designs that are more aerodynamic, capable of directing airflow to minimize turbulent flow effectively.

This research matchs with prior research conducted by Abdel-Salam and Chen, who performed a numerical analysis of airflow around a cargo van model using CFD (Abdel-Salam and Chen, 2014). Their findings revealed the presence of a significant wake region behind the van model, while this current study also highlights changes in drag force on a box vehicle with variations in Wind Deflector design. By comparing the findings of both studies, it can be explained that the additional design element, namely the Wind Deflector, has an impact on enhancing the aerodynamic performance of the box vehicle.

Furthermore, the findings of this study are connected to relevant previous research by Xu and Zhou, focusing on the analysis of airflow around trucks and the development of additional drag reduction devices to optimize aerodynamic performance (Xu and Zhou, 2018). The discovery that the addition of Wind Deflector devices results in a significant reduction in drag force aligns with earlier research findings that indicate drag reduction devices on trucks have a positive effect on reducing aerodynamic resistance.

Hence, this study significantly contributes to understanding the role of the Wind Deflector in the aerodynamic performance of cargo vans. Observations and analyses reveal that variations in Wind Deflector designs directly influence the airflow around the cargo van body, with Wind Deflector Model 3 emerging as a more aerodynamic choice. The installation of the Wind Deflector device can be considered an effective solution for reducing aerodynamic drag on cargo vans.

4. CONCLUSION

In summary, the comprehensive analysis of CFD simulation results in this study reveals a substantial enhancement in the aerodynamic performance of cargo vans with the installation of Wind Deflectors. Notably, Wind Deflector Model 3 stands out as the most effective design, showcasing a significant reduction of 30.30% in drag force and 30.18% in drag coefficient compared to the original configuration. The systematic examination of drag force, drag coefficient, and airflow patterns underscores the valuable contribution of Wind Deflectors in minimizing aerodynamic resistance and improving the overall efficiency of cargo vans.

Looking ahead, it is recommended to pursue further research and development in the realm of aerodynamic enhancements for cargo vehicles. Exploring additional Wind Deflector designs with optimized shapes and orientations could lead to even more significant improvements. The success of Wind Deflector Model 3 suggests the feasibility of broader applications across diverse vehicle types. To validate and refine the simulation results, physical wind tunnel testing is advisable, providing empirical data for a more robust understanding of the practical implications of Wind Deflector installations.

ACKNOWLEDGMENTS

The author on this occasion would like to thank the Kemendikbudristek for providing research funding assistance through the BIMA GRANT for the 2023 fiscal year with contract number 312/F.03.07/2023.

REFERENCES

Abdel-Salam, T. and Chen, D. (2014) 'A Computational Study of Cargo Van Aerodynamics', in *International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics. the 10th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics*, Orlando, Florida: International Centre for Heat and Mass (ICHMT), pp. 2239–2244.

Alawadhi, E.M. (2015) 'Meshing guide', in *Finite Element Simulations Using ANSYS*. 2nd edn. CRC Press. [Print].

ANSYS (2023) 'Ansys Student Software'. Available at: https://www.ansys.com/academic/students/ansys-student (Accessed: 29 December 2024).

Ariyansah, R. (2019) 'Modifikasi Desain Chasis Kendaraan Hybrid Pada Bus Scania K360LB', *Teknobiz : Jurnal Ilmiah Program Studi Magister Teknik Mesin*, 7(3), pp. 156–165.

Banga, S. and Zunaid, Md. (2017) 'CFD Simulation of Flow around External Vehicle: Ahmed Body', *IOSR Journal of Mechanical and Civil Engineering*, 12(04), pp. 87–94.

Gamayel, A. and Octavianus, G. (2021) *Tutorial Ansys Workbench untuk Bidang Mekanikal*. 2nd edn. Bandung, Indonesia: Media Sains Indonesia. [Print].

Hariram, A. *et al.* (2021) 'A Study in Options to Improve Aerodynamic Profile of Heavy-Duty Vehicles in Europe', *Sustainability*, 11(19), p. 5519.

Khosravi, M. *et al.* (2015) 'Aerodynamic Drag Reduction of Heavy Vehicles Using Append Devices by CFD Analysis', *Journal of Central South University*, 22(12), pp. 4645–4652.

Kshirsagar, V. and Chopade, J.V. (2018) 'Aerodynamics of High Performance Vehicles', *International Research Journal of Engineering and Technology (IRJET)*, 05(03), pp. 2182–2186.

Li, Y. *et al.* (2023) 'A Review on Numerical Simulation Based on CFD Technology of Aerodynamic Characteristics of Straight-Bladed Vertical Axis Wind Turbines', *Energy Reports*, 9, pp. 4360–4379.

McTavish, S. and McAuliffe, B. (2021) 'Improved Aerodynamic Fuel Savings Predictions for Heavy-Duty Vehicles Using Route-Specific Wind Simulations', *Journal of Wind Engineering and Industrial Aerodynamics*, 210, p. 104528.

Moriaand, H. and Alam, F. (2012) 'A Computational Simulation of Aerodynamic Drag Reductions for Heavy Commercial Vehicles', in *18th Australasian Fluid Mechanics Conference. 18th Australasian Fluid Mechanics Conference*, Launceston, Australia, pp. 1–4.

Nath, D.S. *et al.* (2021) 'Drag Reduction by Application of Aerodynamic Devices in A Race Car', *Advances in Aerodynamics*, 3(1), p. 4.

Paminto, A.K. (2020) 'Analisis Dan Proyeksi Kebutuhan Energi Sektor Transportasi Di Indonesia', *Jurnal Energi dan Lingkungan*, 16(2), pp. 51–54.

Tu, J. *et al.* (2023) *Computational Fluid Dynamics: A Practical Approach*. India: Elsevier. [Print].

Varadarajoo, P.K. *et al.* (2022) 'Aerodynamic Analysis on the Effects of Frontal Deflector on a Truck by using Ansys Software', *International Journal of Integrated Engineering*, 14(6), pp. 77–87.

Xu, J. and Zhou, S. (2018) 'Flow Field Analysis of Trucks and a Design of an Addtional Drag Reduction Device', *Engineering Review*, pp. 70–78.

Yulianto, F.A., Ariyansah, R. and Octavianus, G. (2023) 'Analisis Modifikasi Desain pada Lokomotif CC-201 Tipe GE U18C dengan Metode Simulasi CFD', *Jurnal Asiimetrik: Jurnal Ilmiah Rekayasa dan Inovasi*, 5(2), pp. 161–170.