New Design Strategies for Truck Forebody Aerodynamics: A Pressure Contour and Velocity Streamline Analysis

Goh Ching Pang

Faculty of Computing and Information Technology, Tunku Abdul Rahman University of Management and Technology, Malaysia

Email: gohcp@tarc.edu.my

Received: *30 November 2023;* ***Accepted:*** *14 December 2023;* ***Published:*** *15 December 2023*

Abstract: The research focuses on the application of aerodynamics beyond aviation, particularly in heavy vehicle design. Traditionally associated with flight, aerodynamic principles play a vital role in minimizing drag for vehicles and trailers, as well as assessing wind stresses on various structures. Heavy vehicles, such as tractor-trailers and buses, consume a significant amount of fuel, and a substantial portion of this consumption is attributed to overcoming aerodynamic drag. The drag force acting on these vehicles increases fuel consumption, making it imperative to minimize drag to enhance fuel efficiency and reduce operational costs. Additionally, reducing drag can lead to improved stability, handling, and safety of heavy vehicles, further emphasizing the significance of aerodynamics in this context. The study provides a comprehensive overview of heavy vehicle aerodynamics, exploring the use of flow-control devices to reduce drag. Computational fluid dynamics (CFD) is employed to simulate the flow field around trucks, using a steady-state formula to evaluate the software's effectiveness in modeling contemporary truck aerodynamics. The primary objective is to enhance the aerodynamic profile of a truck's front end and reduce drag resistance through the implementation of an appropriate drag reduction system. This paper introduces five new designs to compare the drag coefficient, drag force, and fuel consumption with the benchmark design. The results show that Design 1 performs the best among all. The parameters used for comparison include the drag coefficient, drag force, and fuel consumption, which were analyzed using computational fluid dynamics simulations and steady-state formula evaluations. Furthermore, the study discusses future directions for advancing the field, emphasizing ongoing efforts to improve heavy vehicle aerodynamics and the broader implications for the transportation industry.

Keywords: Aerodynamics; ANSYS; CFD; Truck

Introduction

Heavy trucks are commonly employed in construction activities both on roads and at construction sites (Seungwon et al., 2017; Watanabe et al., 2022). However, their performance often suffers due to various factors, including the subpar condition of the roads they traverse, frequent starts and stops, heavy loads, intricate external loads, as well as demands placed on braking and steering mechanisms. Historically, manufacturers primarily focused on enhancing the trucks' durability and rigidity, along with their individual components, to meet a myriad of requirements, paying little attention to the aerodynamic characteristics and design of these vehicles. The aerodynamic drag, which impacts fuel consumption significantly, was often overlooked. To enhance the efficiency of heavy trucks, it is imperative to minimize aerodynamic resistance (Seungwon et al., 2017; Watanabe et al., 2022). As part of the development process, forecasting and improving aerodynamics are essential.

One of the main concerns in heavy vehicle research has always been reducing drag, as it directly correlates with power savings. At high speeds, a significant portion of a heavy vehicle's fuel consumption is attributed to losses in engine performance, aerodynamic losses, rolling resistance losses, and other miscellaneous losses. The sharp surge in fuel prices and the depletion of oil resources have underscored the urgency of addressing this issue. Over the past few years, extensive research has been conducted to develop new flow control techniques and enhance existing ones for drag reduction Watanabe et al., 2022; Mohamed-Kassim & Filippone, 2010). Furthermore, there is now a deeper understanding of the aerodynamics of heavy vehicles. The primary approach to reducing drag in heavy trucks involves manipulating or mitigating flow mechanisms that contribute to aerodynamic resistance, such as minimizing undesirable air separation from the vehicle's outer surface. Truck forebody aerodynamics offers various avenues for enhancing efficiency and minimizing drag.

One notable approach involves employing numerical simulations to assess the impact of different drag reduction mechanisms on a truck's aerodynamic performance. For instance, a comprehensive investigation utilizing Ansys Fluent CFD software was conducted (Ghurri, 2023). This study compared the drag coefficient of a truck fitted with various types of cylinders against one equipped with a conventional windshield. The simulation outcomes demonstrated that opting for specific cylinder designs, in lieu of a traditional windshield, can yield a substantial reduction in drag coefficient and subsequently lead to fuel savings. Another valuable application involves employing Large Eddy Simulations (LES) to examine how various testing parameters affect truck aerodynamic coefficients when subjected to crosswind conditions. In a study by Alejandro et al., (2018). LES was employed to dissect the airflow patterns surrounding both the truck and its surrounding infrastructure. This analysis facilitated the identification of the optimal truck positioning and infrastructure configuration, ultimately contributing to enhanced vehicle stability. In this research, aerodynamic drag and airflow patterns will be investigated. This paper will introduce devices aimed at reducing drag on the front end of the vehicle. Multiple deflectors will be designed and tested to determine the most effective solution. A significant oversight lies in neglecting the impact of aerodynamics on heavy trucks, particularly the role of drag resistance in fuel consumption. To enhance truck efficiency, it is imperative to minimize aerodynamic drag, necessitating a thorough consideration of aerodynamics in the development process. In the context of construction development, heavy trucks are extensively used in Malaysia and abroad, often bearing substantial loads of up to 16 tons in Malaysia and 20 tons in China. This substantial load contributes to increased fuel consumption and environmental pollution. To address environmental concerns and promote energy conservation, improving the fuel economy of trucks by reducing air resistance becomes paramount. The primary objective of this final year project is to devise an effective deflector design capable of minimizing drag force on the truck's forebody. To achieve this overarching goal, several specific objectives will be pursued:

- Modify Truck's Deflector: Implement design modifications to the truck's deflector system with the specific aim of reducing drag force during operation.
- Validate and Analyze Designs: Validate and analyze the modified truck's forebody designs, assessing their aerodynamic performance based on established principles and criteria.

By addressing these objectives, the research endeavors to contribute to the optimization of heavy truck aerodynamics, thereby enhancing fuel efficiency and aligning with contemporary environmental and energy conservation imperatives. As the global emphasis on sustainable practices grows, the outcomes of this research hold potential benefits for both the trucking industry and broader environmental conservation efforts.

Literature Review

Aerodynamic Drag Reduction

The authors (Tang, 2015) highlights the significant role played by digital modeling in the aerodynamic development of new heavy trucks from Dongfeng. The primary parasitic loss in a conventional tractor-trailer configuration operating at highway speeds is drag resistance of aerodynamics. Effective methods for reducing aerodynamic drag in this context offer a cost-effective means to enhance fuel economy.

In another study, researchers concentrated on the tractor system to reduce aerodynamic drag. Their paper examines the impact of several practical add-on methods for reducing drag in tractor-trailer combinations, specifically focusing on the trailer wake, tractor-trailer gap, and trailer sides (Schoon, & Pan, 2007). Another authors discuss reducing fuel consumption for large heavy trucks by optimizing truck design to minimize aerodynamic resistance (Mccallen et al., 2007). Their research led to the development of drag reduction devices designed to intelligently mitigate drag-inducing flow patterns. Environmental and energy concerns have led to heightened regulations on gas consumption and exhaust emissions worldwide. In the pursuit of better aerodynamic drag reduction, understanding the base flow field is crucial for the success of continuous blowing techniques (Baek & Lee, 2020). Kevin Cooper's research aims to employ contemporary computational fluid dynamics codes to optimize tractor-trailer structures. The objective is to identify a robust configuration to reduce this component of aerodynamic drag (Cooper, 2003).

In the study of aerosol dynamics, aerodynamic drag plays a significant role. It influences processes such as impaction, coagulation, diffusion, electric field migration, and sedimentation. Drag force measurements were conducted for various particle classes, including nonvolatile liquid droplets, evaporating droplets, solid spherical particles, and nonspherical solids, revealing insights into their behavior (Davis et al., 1987). Historically, the aerodynamic drag of road vehicles was not considered significantly significant. However, recent research projects, such as those conducted by AeroVironment Inc., have focused on truck drag reduction (Bayındırlı, 2016). Additionally, surface pressure and drag measurements for a 1/32 scale truck and trailer mounted in a wind tunnel yielded a drag coefficient (CD) of 0.704 for the combination of truck and trailer (Choi, 2013). For aerodynamic drag reduction, researchers have explored three primary areas: the tractor-trailer gap, wheels/underbody, and trailer base. Introducing a low-speed bleeding flow into the trailer wake and tractor-trailer gap can alleviate aerodynamic drag resulting from flow separation at the trailer base and cross-flow in the gap, effectively reducing flow velocity in the underbody area.

Forebody Drag Reduction

This section focuses on drag reduction devices for heavy vehicles, primarily tractor-trailers. Although experiments and research on drag reduction for simpler vehicle types are also discussed, the primary emphasis is on forebody drag reduction, depending on their application locations on the vehicle (Choi, 2013). Researchers worldwide have made numerous efforts to reduce forebody drag, primarily in the context of heavy trucks. The forebody, encompassing the distance from the trailer to the front of a heavy vehicle, represents nearly 45% of drag reduction in conventional heavy trucks on highways. Various patterns have been designed to control airflow around the truck's forebody, with a specific focus on cab deflectors in this paper.

Deflector Effect

Deflectors, such as plates or attachments for redirecting airflow, are instrumental in reducing drag. The deflector's orientation is described by the angle between the deflector and the top of the box (H1−H2). Altering the deflector angle from 5° to 35° affects the drag coefficient. The optimal deflector angle was found to be 19.6°, as deviations from this angle either increased drag due to airflow disruption or drag due to pressure distribution. Thus, 19.6° was determined to be the optimal angle relative to the defined height difference. This discovery has implications for the reduction of drag in heavy vehicles (Choi, 2013).

Computational Fluid Dynamic (CFD)

A critical aspect of modern vehicle design involves the application of computational fluid dynamics (CFD) to analyze and assess the aerodynamic properties of various vehicle components, including the chassis of heavy trucks. This computational approach enables researchers to study complex vehicle shapes and make informed design decisions (Nishikawa, 1989). CFD simulations involve dividing the physical area into small finite volume elements and numerically solving governing equations that describe fluid behavior. These simulations enable the evaluation of design models at different speeds. The solver iterates through a series of calculations for mass and momentum at each node of the profile to determine flow behavior. In the final stage, flow across the vehicle, pressure contours, velocity contours, and turbulence kinetic energy contours are analyzed. Graphs are generated to depict the drag coefficient values for the vehicles.

Methodology

TRIZ methodology that will be applied in this research as shown in Figure 1. The CAD drawing of the truck with deflector are created by using Fusion360 as shown in Figure 2. Then the file will be exported to STEP file. The STEP file will then imported into ANSYS for simulation study. In this research, five deferent designs of the truck will be drawn and compared using ANSYS. Also, the drag force will be compared using between ANSYS and following equation (Maxemow, 2013):

$$F_D = \frac{1}{2} p v^2 c_D A \text{................................ (equation 1)}$$

where Density of fluid, p =1.225; speed of the object relative to the fluid, v =25m/s; and cross sectional area, A= 8.4985m^2

The details of the designs are shown in Table 1. Table 1 displays the fillet radius for each design, encompassing the left, right, and top edges of the truck deflector. Regarding the left and right edges, each fillet radius increments by 50mm compared to the subsequent design. Consequently, the fillet radius for designs 1 through 5 is 250mm, 300mm, 350mm, 400mm, and 450mm, respectively. Similarly, for the top edge, the increment between each design is 20mm. Therefore, the fillet radius for the top edge of designs 1 through 5 is 220mm, 240mm, 260mm, 280mm, and 300mm, respectively. It is noteworthy that the height of the deflector remains constant at 1400mm for all designs. The experimental parameters are shown in Table 2.

Next, Computational Fluid Dynamics (CFD) will be applied to evaluate the aerodynamics of each design, specifically examining the streamlines of the deflector on the truck's forebody. This analysis aims to identify any potential issues within the design. Additionally, wind tunnel experiments will be performed on a heavy-duty truck featuring a previously designed deflector system, and these experiments will utilize CFD simulations. To enhance the aerodynamic efficiency of trucks, it is essential to comprehend the flow field over them. Computational Fluid Dynamics (CFD) was employed to simulate the flow field over a truck, utilizing a steady-state formulation. The primary objective of this simulation was to assess the capabilities of the current CFD software used in the construction of aerodynamic vehicles, particularly trucks. This task aims to reduce the drag coefficient by refining the aerodynamic profile of the tractor, thereby enhancing both the fuel efficiency of the truck and the overall operational efficiency of the vehicle. Moreover, as a body moves through a stationary fluid or when fluid flows past a body, it imparts a resultant force known as drag force. This force acts parallel to the velocity of the free stream, and understanding and minimizing it are crucial for optimizing the aerodynamic performance of trucks.

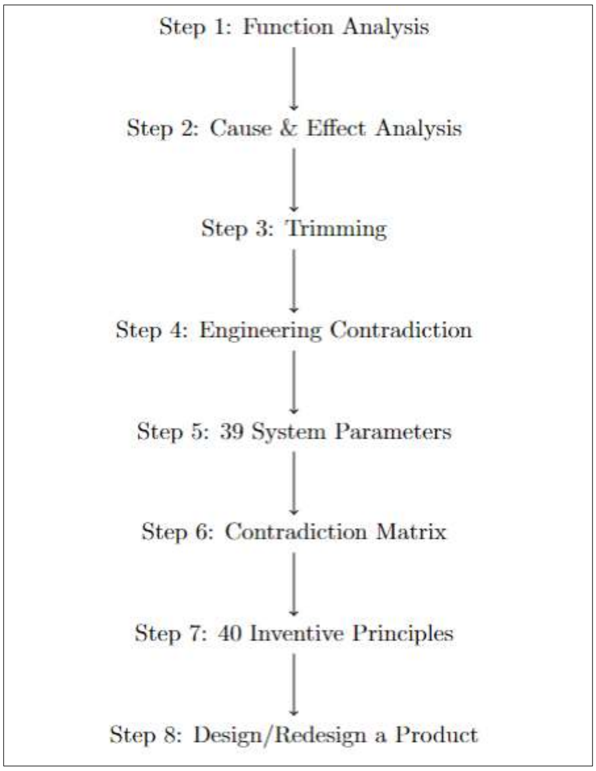


Figure 1: Flowchart of the process

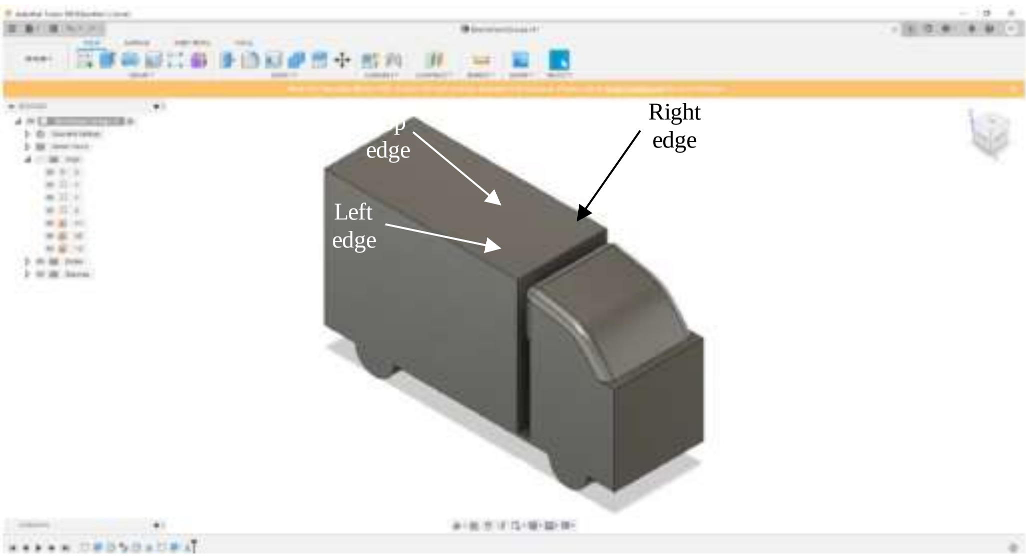


Figure 2: CAD drawing of truck

Table 1: Dimension of five designs

Deflector Dimension (mm)	Design 1	Design 2	Design 3	Design 4	Design 5
Left Edge	250	300	350	400	400
Right Edge	250	300	350	400	450
Top Edge	220	240	260	280	300
Height	1400	1400	1400	1400	1400

Table 2: Improving and worsening parameters

	Parameter	Reason
Improving parameter	3 Length of moving object	Length or angle of the deflector may need to change
	9 Speed	The truck can accelerate with low fuel consumption
	10 Force	Force=mass x acceleration
	12 Shape	The appearance of truck is ease to air resistance
Worsening Parameter	34 Ease of repair	Cannot be repaired
	39 Productivity	No productivity

Result & Discussion

Analyzing the static pressure contours from Figure 2 provides a clear understanding of the aerodynamic characteristics of each design, including the benchmark. The sample simulation result of Design 1 is shown in Figure 3(a). The benchmark design exhibits the highest maximum static pressure at 456.6 units, a figure that suggests a robust aerodynamic profile. This elevated pressure implies that the benchmark design may excel in scenarios that demand high-speed performance or high load-bearing capacity. Designs 1 and 3, with maximum pressures of 448.6 and 448.5 units respectively, are closely aligned with the benchmark, albeit with a slightly less robust profile. This marginally lower maximum pressure could indicate a more versatile operational range, possibly offering better performance across a variety of conditions.

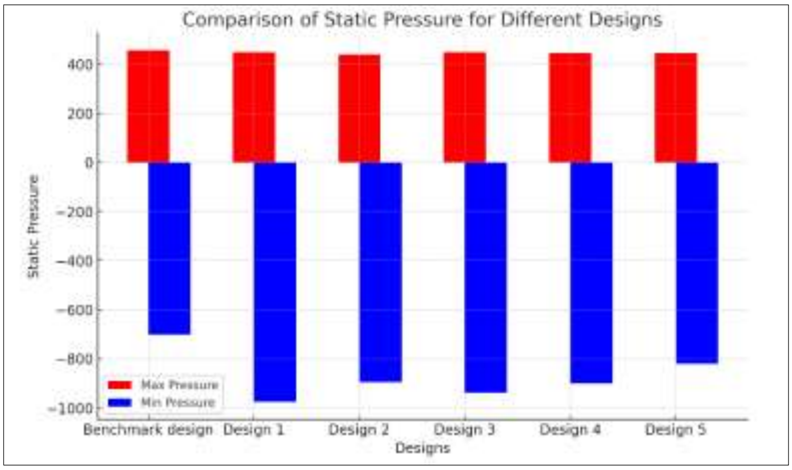
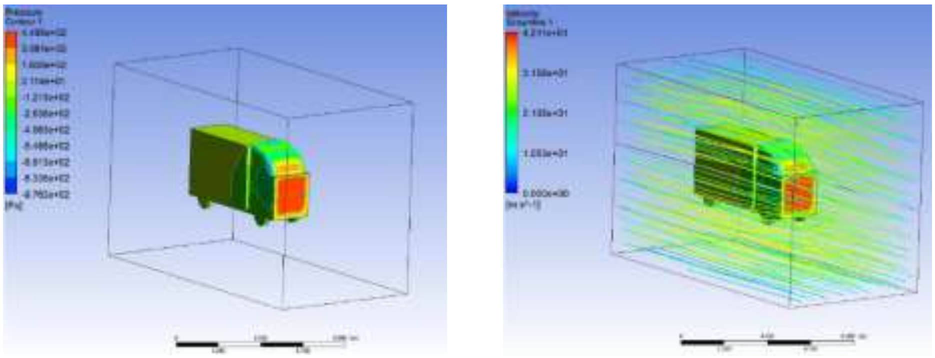


Figure 2 Static pressure comparison between 5 designs



(a) (b)

Figure 3: Simulation result of (a) pressure contour (b) velocity streamline

Designs 2 and 4, on the other hand, show the lowest maximum pressures at 440.6 and 445.0 units, respectively. These lower pressures hint at a different design focus, potentially prioritizing efficiency or performance in conditions where lower aerodynamic resistance is beneficial. On the flip side, the minimum static pressures paint a contrasting picture. Design 1 is particularly noteworthy with the lowest minimum pressure at -976.2 units, suggesting a wide operational pressure range. This extensive range indicates potential suitability for dynamic scenarios, such as those requiring rapid acceleration or maneuverability. In contrast, the benchmark design maintains a higher minimum pressure at -701.8 units, pointing towards a more stable and constrained operational range, which might favor consistent performance across a spectrum of conditions. The disparity in pressure ranges across these designs is telling. The benchmark design's narrower range suggests stability and predictability in performance, which could be crucial in certain operational scenarios. Design 1's broad pressure range signifies its aptness for high dynamic performance, albeit potentially at the cost of stability. The other designs, with intermediate pressure ranges, balance between dynamic performance and stability, each catering to specific operational needs.

In summary, the benchmark design appears optimal for scenarios demanding stability and predictable performance. Design 1, with its broader pressure range, is more aligned with high dynamic performance scenarios, like those requiring agility. Designs 2, 3, and 4, present a balance, making them adaptable to a range of applications, depending on specific aerodynamic requirements. The choice among these designs should be guided by specific performance criteria, where static pressure characteristics play a pivotal role.

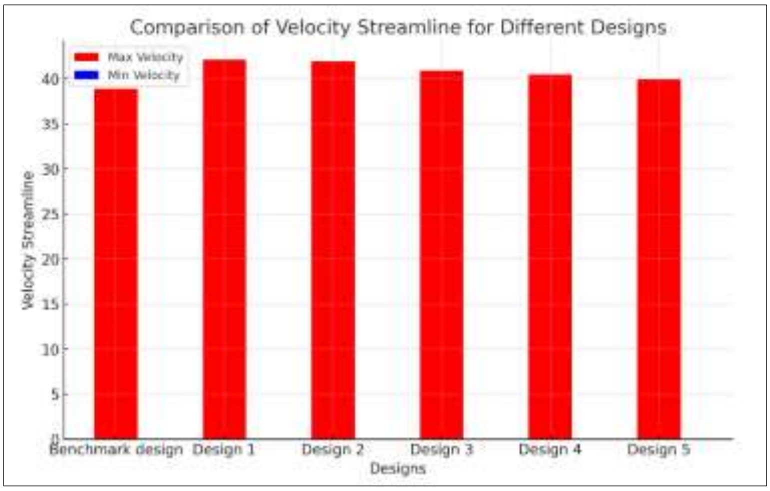


Figure 4: Comparison of velocity streamline between five designs

The analysis of the velocity streamline data from Figure 4 reveals distinct performance characteristics for the five designs and the benchmark in terms of their maximum and minimum velocity streamline values. The sample simulation result of Design 1 is shown in Figure 3(b). The benchmark design, with the lowest maximum velocity streamline value at 38.9, indicates a potential optimization for lower speed applications or a focus on higher stability at reduced velocities. This suggests that the benchmark design might excel in applications where precision and control at lower speeds are critical, possibly at the expense of top-end speed performance. In contrast, Design 1, with the highest maximum velocity streamline value at 42.1, appears to be optimized for higher speed applications. This higher value implies a design focus on maximizing speed, which could be beneficial in scenarios where rapid movement is crucial. However, this might come at the cost of reduced stability or lower efficiency at slower speeds, a trade-off that needs to be considered for specific applications. It's particularly noteworthy that all designs, including the benchmark, share a minimum velocity streamline value of zero. This uniformity suggests a common baseline in operational performance across all designs, indicating a similar point at which each design ceases to function optimally or enters a state of minimal operational activity. The variation in the maximum velocity streamline values is the key determinant in the suitability of each design for different applications. Designs with higher maximum values, such as Design 1, are likely more suited for high-speed scenarios where agility and rapid movement are prioritized. Conversely, designs with lower maximum values, like the benchmark, might be better aligned with applications prioritizing stability, control, and efficiency at lower operational speeds. In summary, the decision to select a particular design from these options should be based on the specific velocity requirements of the intended application. The velocity streamline profiles provide valuable insights into each design's operational strengths and weaknesses, enabling a more informed selection process that weighs the needs for speed, stability, and efficiency in different operational contexts.

Table 3: Comparison between five designs

Design	Drag Coefficient	Drag Force by ANSYS (N)	Drag Force by equation (N)	Fuel Consumption (Liters)
Benchmark Design	0.8173	2831.87	2705.68	21.78
Design 1	0.7997	2704.76	2601.69	20.95
Design 2	0.8472	2861.41	2756.22	22.19
Design 3	0.8467	2855.63	2754.60	22.18
Design 4	0.8581	2889.95	2791.68	22.48
Design 5	0.8482	2852.29	2759.48	22.22

The results for drag force and drag coefficient are indicative of fuel consumption. This assessment aimed to calculate the drag force associated with various designs. The variables manipulated included the top edge fillet radius, the side edge fillet radii, and the height of the deflector. Meanwhile, the material (aluminum) and the dimensions of the deflector were kept constant. The outcomes measured were the drag force, pressure contour, and velocity streamline. Based on Table 3, the most optimal drag coefficient and drag force were determined. Upon calculating the Reynolds number, it was found that all designs fall under turbulent flow, attributed to the high velocity of the truck. A Reynolds number over 4000 signifies turbulent flow. The use of deflectors has been shown to improve the aerodynamic characteristics of the truck trailer, reducing both the drag coefficient and drag force. Among all the designs, including the benchmark, Design 1 achieved the most optimal fillet radius. Its drag coefficient for the truck was significantly lower than that of the benchmark design, indicating reduced drag force and, consequently, greater fuel savings. Flow visualization demonstrates the flow structure around the truck trailer, confirming turbulent flow in Design 1. While turbulent flow does not channel wind as smoothly as laminar flow across the truck's container, it is a result of the high velocity. Despite the turbulent nature of all designs, Design 1 effectively reduces drag force. Table 3 shows that the benchmark design has a drag coefficient of 0.8173. Throughout the designs, Design 4 has the highest drag coefficient, followed by Designs 5, 2, and 3, with coefficients of 0.8482, 0.8581, 0.8472, and 0.8467, respectively. Design 1 exhibits the lowest and most favorable drag coefficient, contributing significantly to aerodynamic drag reduction. The benchmark design's drag force, as obtained by ANSYS simulation and equation calculation, is 2831.87N and 2705.68N, respectively. Design 4 exhibits the highest drag force in both ANSYS simulation and equation calculation, with values of 2889.95N and 2791.68N. This is followed by Designs 2, 3, and 5. The lowest drag force is observed in Design 1, with 2704.76N in the ANSYS simulation and 2601.69N in equation calculation. In conclusion, based on both ANSYS simulations and equation calculations, Design 1 is the most optimal, offering the maximum potential for fuel consumption savings.

Conclusions

The conclusion of the research paper highlights the categorization of drag reduction systems for heavy trucks into under-vehicle, foundation, and forebody systems. By implementing appropriate drag reduction devices, heavy vehicles can effectively minimize aerodynamic drag, resulting in significant fuel savings—a primary goal of this project. The research successfully achieved its objectives by focusing on designing an efficient deflector to reduce drag on the truck's forebody. Utilizing the TRIZ methodology, the research determined the aerodynamic variables and parameters crucial for deflector design. Subsequently, deflector designs were created, validated, and analyzed based on their aerodynamic effectiveness. The optimal design, labeled as Design 1, emerged as the most effective in reducing drag force. The research recommends that every heavy truck be equipped with a substantial and efficient deflector to mitigate wind and air resistance. Recognizing the variability in heavy trucks' requirements, it suggests tailoring deflector shapes, sizes, and designs to suit each truck's specifications. Additionally, employing advanced software for evaluating and analyzing aerodynamic coefficients during deflector design is advised. To enhance design further, a comprehensive understanding of fluid mechanics, particularly in drag reduction methods, is crucial. This research not only successfully achieved its primary objective of designing an efficient deflector for drag reduction but also emphasized the importance of tailored solutions for diverse heavy truck configurations. The recommendation underscores the need for advanced software and a deep understanding of fluid mechanics to continually improve aerodynamic designs for heavy trucks, emphasizing the continuous improvement of aerodynamic designs for heavy trucks.

Acknowledgement

The researcher did not receive any funding for this study, and the results have not been published in any other sources.

References

Alonso-Estébanez, A., del Coz Díaz, J. J., Álvarez Rabanal, F. P., Pascual-Muñoz, P., & García Nieto, P. J. (2018). Numerical investigation of truck aerodynamics on several classes of infrastructures. Wind and Structures, 2*(1), 35. https://doi.org/10.12989/WAS.2018.26.1.035

Baek, S.-W., & Lee, S. (2020). Aerodynamic drag reduction on a realistic vehicle using continuous blowing. Microsystem Technologies, 26. https://doi.org/10.1007/s00542-019-04355-w

Bayındırlı, C., Akansu, Y., & Salman, M. (2016). The determination of aerodynamic drag coefficient of truck and trailer model by wind tunnel tests. International Journal of Automotive Engineering and Technologies, 5, 53. https://doi.org/10.18245/ijaet.11754

Choi, H., Lee, J., & Park, H. (2013). Aerodynamics of heavy vehicles. Annual Review of Fluid Mechanics, 46. https://doi.org/10.1146/annurev-fluid-011212-140616

Cooper, K. (2003). Truck aerodynamics reborn - lessons from the past. https://doi.org/10.4271/2003-01-3376

Davis, E. J., Zhang, S., Fulton, J., & Periasamy, R. (1987). Measurement of the aerodynamic drag force on single aerosol particles. Aerosol Science and Technology, 6, 273–287. https://doi.org/10.1080/02786828708959139

Ghurri, A. (2023). CFD simulation of aerodynamics truck using cylinder as drag reduction device. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences. https://doi.org/10.37934/arfmts.105.2.166181

Maxemow, S. (2013). That's a drag: The effects of drag forces. Undergraduate Journal of Mathematical Modeling: One + Two, 2. https://doi.org/10.5038/2326-3652.2.1.4

Mccallen, R., Salari, K., Ortega, J., Castellucci, P., Pointer, W., Browand, F., Ross, J., & Storms, B. (2007). DOE project on heavy vehicle aerodynamic drag. https://doi.org/10.2172/1036846

Mohamed-Kassim, Z., & Filippone, A. (2010). Fuel savings on a heavy vehicle via aerodynamic drag reduction. Transportation Research Part D: Transport and Environment, 15(5), 275–284. https://doi.org/10.1016/j.trd.2010.02.010

Nishikawa, Y., Watanabe, Y., & Hideki, N. (1989). A review of Japanese albacore fisheries in the Pacific and Indian Oceans [WP10]. Noumea, New Caledonia: South Pacific Commission. South Pacific Albacore Research Workshop, Suva, Fiji, 14 - 16 June 1989. 13 p.

Schoon, R., & Pan, F. (2007). Practical devices for heavy truck aerodynamic drag reduction. SAE Technical Paper 2007-01-1781. https://doi.org/10.4271/2007-01-1781

Seungwon, P., Dong, L., Chulhaeng, H., Byoeunggyu, Y., & Daewook, K. (2017). An experimental study on GHG emissions reduction and fuel economy improvement of heavy-duty trucks by using aerodynamics device package. Transactions of the Korean Society of Automotive Engineers, 25(2), 207-218.

Tang, K., He, L., Zhao, Y., Friz, H., & Li, B. (2015). The aerodynamic development of a new Dongfeng heavy truck. SAE Technical Paper 2015-01-2886. https://doi.org/10.4271/2015-01-2886

Watanabe, T., Shibata, M., Fukaya, N., Nagai, T., & Suzuki, T. (2022). Setting development targets for fuel cells and systems for heavy-duty trucks using a comprehensive model-based approach. ECS Meeting Abstracts, MA2022-02(40), 1468. https://doi.org/10.1149/MA2022-02401468mtgabs