EXPERIMENTAL INVESTIGATION ON CYLINDER AND ITS POSITIONING AS A DRAG REDUCER ON CONTAINER LORRIES

Article history
Received
22nd February 2024
Received in revised form
29th April 2024
Accepted
8th May 2024
Published
1st June 2024

Muhammad Azreef Hilman Mohd Rizalman and Azmin Shakrine Mohd Rafie*

Department of Aerospace Engineering, Universiti Putra Malaysia, 43400 Serdang, Selangor, Malaysia

*Corresponding author
shakrine@upm.edu.my

ABSTRACT

The most common heavy-duty vehicles for moving huge quantities of goods in both urban and rural locations are container lorries. The number of hazardous pollutants in the environment has dramatically increased as a result of the huge capacity engines on this vehicle. To solve, it makes more sense to think about enhancing vehicle design to boost aerodynamic efficiency as opposed to other worries. However, it is impossible to make significant changes to these heavy-duty vehicles without influencing how much cargo space they can hold since the original container shape is standardized worldwide. Therefore, a cylinder is proposed to be fitted onto the lorry to reduce the drag force acting on the container lorries. Thus, this study is focused on experimenting on the effects of a static cylinder that blends in with an existing generic container lorry model, the Mercedes Benz Actros together with a roof fairing with fenders and aims to reduce its drag force by finding the right position for the cylinder. In this research work, a study is conducted using the wind tunnel to prove the effectiveness of cylinders in reducing drag of container lorries by different cylinder position configurations. According to the experiment, Cylinder Position 5 is the best configuration to reduce Cd acting up the lorry with the drag coefficient of 0.931.

KEYWORDS

Container Lorry; Coefficient Drag; Cylinder

INTRODUCTION

The aerodynamic drag of bluff bodies has been investigated in several scientific studies, with a focus on the drag associated with their blunt bases because they are relevant to and have an impact on the aerodynamics of road vehicles. At highway speeds, aerodynamic drag results in a significant parasitic loss in a typical Class 8 line-haul tractor-trailer system. Researchers have significantly improved their understanding of the mechanics underlying drag reduction with blunt-based bluff bodies over the previous few decades, in part because of the rise in fuel prices [1]. Thus, aerodynamic performance has been taken into account in the design of numerous vehicles, but for larger road vehicles, such as container trucks, major design changes are needed to achieve good aerodynamic performance. The vehicle must have a tapered shape to reduce drag pressure in order to have outstanding aerodynamic efficiency, which is highly challenging because it would decrease cargo capacity [1].

In relation to aerodynamics, the phrase "drag coefficient" is generally used in the automotive industry. While the drag force works in the same direction as the fluid flow, the former does not. How a vehicle moves through the air it is in is influenced by its drag coefficient. As a result, different cars will surely have different drag coefficients depending on how they are classified or even what model they are. For instance, a motorcycle has one of the lowest drag coefficients (Cd) of any type of vehicle.

On a motorcycle, however, the rider and passenger have a significant impact on the drag coefficient, unlike covered cars. Depending on the design, a car, multipurpose vehicle (MPV), or sport utility vehicle (SUV) has a Cd that ranges from 0.2 to 0.5. Sport cars are specifically designed to have a low drag coefficient. Some sports cars can reach top speeds with a drag coefficient of 0.1 [2]. While a bus's drag coefficient (Cd) is around 0.7, larger vehicles (over 5 tonnes), such as buses, tractor-trailers, and container lorries, have substantially higher Cd than other forms of transportation. Furthermore, the Cd of the truck head alone may be 0.6, and the Cd of the truck head and trailer together could be 0.8. In order to reduce aerodynamic drag, it is imperative to look at the effectiveness of fuel-saving technology on current vehicles given the enormous increase in the number of trucks on the road throughout the world as a result of expanding logistic transportation.

A cost-effective strategy to reduce aerodynamic drag while improving fuel efficiency and keeping acceptable driving stability is to use practical gadgets in this situation. They are one of the most effective methods available. Heavy trucks have the ability to move at high speeds along interstates and accumulate far too much over time. For the purpose of enhancing vehicle performance, car makers are investing in aerodynamic studies [3]. The drag force generated by the vehicle's aerodynamics affects all aspects of the vehicle's operation, including performance, fuel efficiency, acceleration, handling, noise, and comfort. Additionally, the engine cooling system and the heating interior ventilation system are strongly related to aerodynamics. The aerodynamic drag coefficient rises in proportion to the square of the speed [3]. Heavy commercial trucks are thought to be less aerodynamically efficient than other ground vehicles due to their asymmetrical body shapes. A large commercial truck travelling at 100 km/h requires about 52% of its total gasoline to overcome the aerodynamic drag [4].

Nowadays, most tankers come with a variety of fuel-saving features or accessories that use aerodynamic designs on the front and other parts of the truck to lessen drag. Without changing the truck's projected frontal area, it is possible to modify the vehicle's shapes, including the container box, in a more streamlined way. These external attachments can reduce aerodynamic drag due to the outer shapes, dimensions, and placements [5]. Thus, reducing aerodynamic drag enhances a truck's fuel efficiency while also reducing greenhouse gas emissions [6–8]. So far as vehicle aerodynamics are concerned, drag continues to be the most crucial factor. There hasn't been much in-depth study or documentation of the aerodynamic impacts of the front and side aerodynamic fairing models that are now available, or of their configurations. As a result, minimising aerodynamic drag is a more pertinent issue for heavy trucks, which account for a sizable share of outside-the-city traffic and travel a great distance at high speeds annually [9]. One of the main factors in aerodynamic drag, which is the factor that affects fuel consumption, is base drag. Previous studies have shown that the base drag occurs toward the rear of the vehicle, where the cargo space is located. A vehicle with a bluff body that is moving at a constant speed on a level surface will use up to 40% of its fuel energy to combat aerodynamic drag and 45% to fight rolling resistance, leaving only 5% of its fuel energy to power the gears and other losses [10]. The passive and active flow control methods are just a couple of the techniques that can be employed to enhance the aerodynamics of the vehicles. With three separate variants, it has allegedly improved aerodynamics. [11] With the nose cone, nose cone curvature, and chassis skirt, he increased aerodynamic efficiency by 3%, 4%, and 7%, respectively. With an increase in windscreen attack angle on a commercial vehicle, the drag coefficient decreases [12].

Modi et al. [13] claim that enhanced aerodynamics were attained on a 1/6 model vehicle and trailer.

The front surface area of the trailer has increased by 12.5% and 28% in terms of vertical and horizontal spoilers, respectively, and produces a sizable amount of aerodynamic drag. Gilieron and Kourta [14] increased their aerodynamics by 12% by using a redirector plate, whereas Fouree et al. [15] reduced the drag coefficient by 9% by using flow deflectors based on different deflector angles on a basic vehicle model.

Because of this, the drag-reducing device covered in this article is intended for heavy-duty vehicles, particularly container trucks, due to their bluff bodies as well as its positioning to reduce drag.

Problem Statement

In recent years, heavy-duty trucks have become more common as modes of freight and transportation. Container trucks are the heavy-duty vehicles that are most frequently used to transport massive quantities of products in both urban and rural areas. Large capacity engines on this specific vehicle have significantly increased the number of hazardous emissions in the environment. Basic vehicle design, zone of operation, driving practices, and climate issues can all be improved to lower emissions into the environment. Thoughts of improving vehicle design to increase aerodynamic efficiency are plausible nonetheless. A vehicle can now have its design altered to make it more slimline, whether it be an automobile or a multipurpose vehicle. The truck's capacity to transport freight, however, may be harmed. Because of this, truck manufacturers created a standard design with additional body changes, including rooftop spoilers, to cut down on aerodynamic losses. Most of these body alterations are fairly substantial and don't fit with the current container truck design. It's unusual to find a product that is both lightweight and secure to use.

Truck manufacturers developed a common design with further body alterations, such as rooftop spoilers, in order to reduce aerodynamic losses. Most of these body modifications are quite significant and don't go along with the present container truck design. It is rare to come across a device that is both small and secure to use. Therefore, the aim of this research work is to investigate the effects and use of static cylinders as a drag reduction for heavy-duty vehicles acting as a bluff body. In addition, this study will show how the design might be effective without sacrificing cargo capacity. Through wind tunnel tests, this research will examine and discuss the reduction of bluff body drag at different cylinder locations.

The purpose of the study is to determine whether a drag-reducing device, such as a static cylinder, can be used to lessen the drag effect on a container truck. The study will conduct additional testing on various cylinder locations to see which is most appropriate for achieving the largest decrease percentage in order to better understand its effectiveness.

Aerodynamics of Road Vehicles

The field of study known as aerodynamics investigates how fluids and moving solid things interact. It has a big effect on things like speed, fuel economy, and visual appeal, which are all things that people would view to be luxurious. The term "aerodynamics" is almost always used to refer to the measurement of the drag coefficient, a dimensionless statistic that shows how much a moving body and the surrounding fluid are at odds. Depending on fuel efficiency and speed, the performance of road vehicles increases as the drag coefficient decreases. The following Equation (1) may be used to calculate the drag coefficient:

$$C_d = \frac{2F}{\rho A V^2} \qquad (1)$$

Where Cd is the drag coefficient, F is the drag force, ρ is the air density, A is the frontal cross-sectional area, and v is the vehicle's velocity. Even though this experimental research has been extensively used to study aerodynamics, setting up a reliable experimental setup that takes into account a range of elements like mechanical facilities, operational expenses, and scale flexibility is still a challenging process. As a result, computational fluid dynamics (CFD) has been a popular aerodynamics technique for more than 50 years. For instance, a verified CFD research could reduce the overall number of experimental trials by half or even more. The shape of the body that is engulfed in air and the rate of movement are directly related. Birds, planes, and bullets all have similar body shapes that are intended to minimise drag. A design that enables an object to move effectively inside a fluid-filled zone is called an airfoil, sometimes known as an aerofoil. Manufacturers have employed design patterns based on the shape of the airfoil on the main body and auxiliary equipment to achieve high-speed transportation on the road for more than a century.

Components of Drag

A moving object is influenced by drag in two different ways. Pressure drags, which operates against a surface in the direction of motion as a result of pressure forces acting on the body, is the first type of drag. Friction drag, on the other hand, operates perpendicular to a surface as a result of shear and viscous processes in the flow close to the body surface. Pressure drag is the key factor for large vehicles like tractor-trailer combinations and buses because of the vast surfaces perpendicular to the main flow direction and the significant wake produced by the bluntness of such vehicles' back ends. The strongest pressure forces on a vehicle are those acting on its front and rear faces, as well as those caused by the separation between a tractor and trailer. The vast empty spaces under the bodies of tractor-trailer combinations also help to reduce pressure drag. Furthermore, cooling flows in an automobile's engine compartment are dominated by pressure-drag effects.

Friction drag is present throughout the external surfaces of large vehicles, particularly the sides and tops of buses and trailers, but it only contributes a small amount to total drag (10 percent or less [16]), making it a poor candidate for drag-reduction strategies. Road vehicle aerodynamics is mostly focused on pressure drag, as opposed to air vehicles, which have streamlined bodies that primarily contribute to friction drag. This is why the vehicle's massive body is necessary. The road vehicle and ground transportation industries may not always be affected by our understanding of how to reduce drag on airborne vehicles. Along with saving fuel, reducing drag also offers other benefits including improved aerodynamic stability and reduced splash and spray.

METHODOLOGY

The techniques, methods, and selection utilised to attain the project's goals are described in this chapter. The stages of methodology indicate a project objective and lead to the project's final output.

Container Lorry Selection

The Mercedes Benz Actros vehicle served as inspiration for the design of the container truck. One of the vehicles that is frequently used in Malaysia is the Mercedes Benz Actros. Pemborong Bumijaya Sdn Bhd (PBJ), a major player in East Malaysia's plantation and construction industries, has a fleet of 10 Mercedes-Benz ACTROS prime movers. PBJ is situated in Sabah. Hap Seng Trucks Distribution Sdn Bhd (HSTD), a local distributor for Daimler Trucks, celebrated the handover of a fleet to PBJ through its general manager, Bernard Chun. PBJ had purchased 10 Mercedes-Benz ACTROS vehicles, specifically the 3340S 6x4 with retarder, and this multi-unit purchase marked the first time PBJ had purchased an ACTROS [17].

In addition, Hap Seng Trucks Distribution Sdn Bhd (HSTD) and authorised dealer Hap Seng Commercial Vehicle (HSCV) gave Uniglory Logistics Sdn Bhd 10 new Actros prime movers 3344 brands [18]. The quality of the truck was another factor in the decision to go with the Mercedes Benz Actros. In the day-to-day operations of transportation businesses, the new Actros is an appealing prospect. It has received various accolades from all over the world, including the prestigious and highly regarded "International Truck of the Year 2020." This is the fifth year that an Actros has received the prestigious award, which highlights the fact that the flagship of Mercedes-Benz Trucks continues to set standards for dependability, efficiency, and comfort. The award was chosen by a strong jury of international commercial vehicles journalists from 24 countries. Lastly, the model was chosen due to availability as a diecast in scale 1:20 and on sale at e-commerce websites such as Shopee, Lazada and Carousell Malaysia.

Cylinder Selection

Using the data from Hamdan's earlier experiments as a guide, the cylinder concept was selected [20]. In an experiment employing a spinning cylinder and a BLDC motor, a helical-shaped cylinder was employed, but the findings showed that the drag reduction was just a small percentage, with no meaningful effects. The Savonius Turbine experiment is another subject that has been learned from Hamdan's experiment. The experiment was unsuccessful because the body could not freely spin and the turbines were unable to rotate the cylinder. This was primarily because of the turbines' excessive weight or because of numerous fabrication errors and high tolerance.

Placement of Cylinder

The location of the cylinder is determined by the outcomes of Kamid's and Hamdan's experiments [19] [20]. The majority of the devices were mounted on the back of the container or truck, according to several earlier studies on drag reduction systems. Airflow over and around a moving vehicle is allegedly disrupted as a result. To distribute the air, the manufacturers frequently use the rear spoiler. It reduces the turbulence that a moving car produces. As a result, it helps to lessen air resistance and turbulent flow. In light of this, the new location will likewise be at the back of the container. The spots at the top back of the container, the back corner, and lastly the top back of the container was selected.

Conceptual Design and Drawing

Once the design parameters are set, three types of computer aided design (CAD) software, which are Computer Aided Three-dimensional Interactive Application (CATIA) V5R21, Shapr3D, and Thinkerspace, are used for the designing of the few parts that are involved in this project. Figure 4 shows an isometric view of the roof fairing with fenders of the container lorry as a drag reduction device. The roof fairing design was taken from common current designs in the market. Figure 5 illustrates an isometric view of the design of the cylinder, which acts the same as the latter.

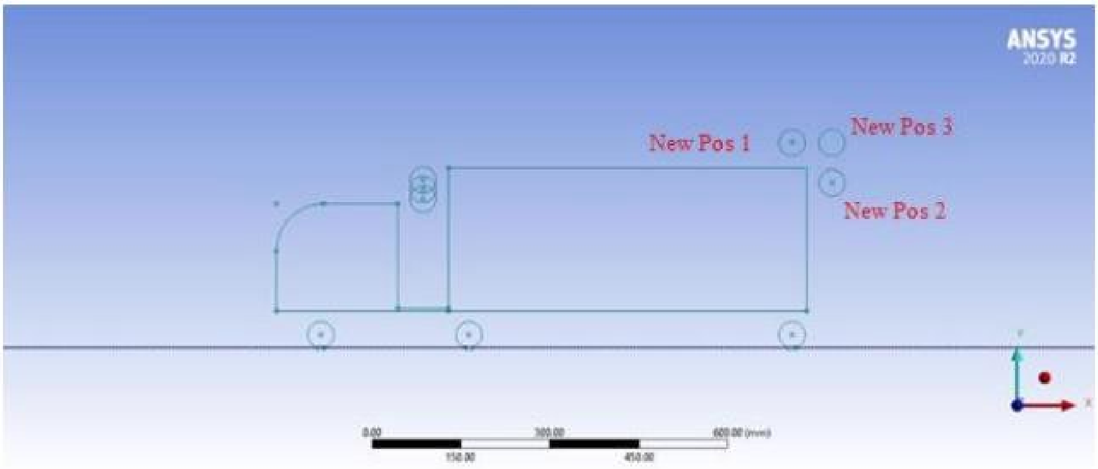

Figure 1: Geometric Model of Lorry with Three New Positions of Cylinder [19]

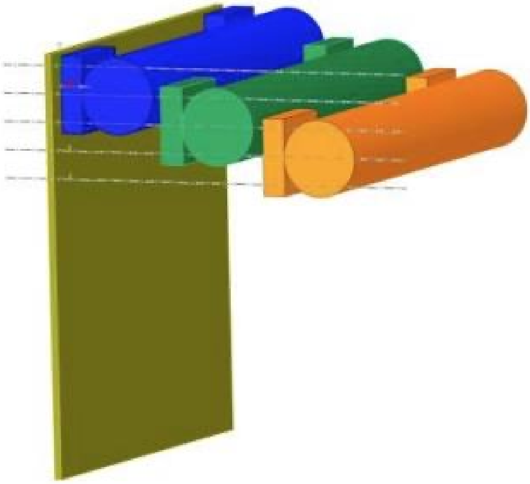

Figure 2: Geometric Model of Lorry with Three Positions of Cylinder [20]

Figure 3: Geometric Model of Lorry with Six Positions of Cylinder [21]

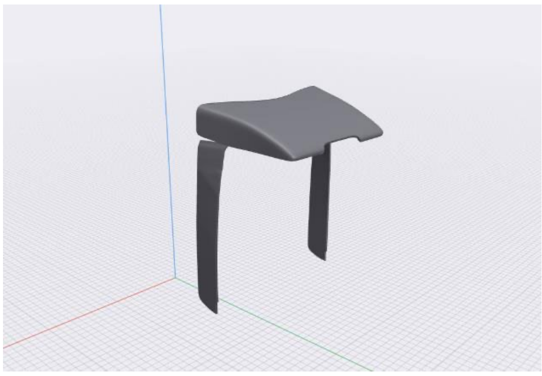

Figure 4: CAD drawing of roof fairing with fenders [21]

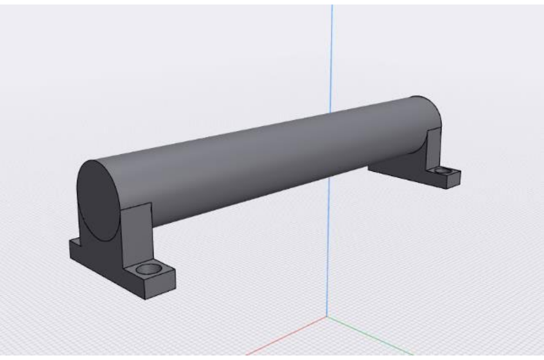

Figure 5: CAD drawing of static cylinder [21]

Testing and Experiment

For the parametric studies, three testing variations were identified as Test 1, which includes the roof fairing with fenders; Test 2, which includes the static cylinder; and Test 3, which includes both the roof fairing with fenders and static cylinder, which were identified earlier during the concept selection in the designing section. The other apparatuses used in this testing are:

a. Wind tunnel (closed end)
b. Container Lorry model (Mercedes Benz Actros)
c. Weighing scale (for calculating drag force)
d. Digital manometer (for calculating wind speed of wind tunnel)

For analysis purposes, several data and assumptions were collected, as shown in the table below:

Table 1: Data predetermined prior to testing [21]

Data	Value
Air Density	1.126 – 1.151 kg/m^3
Air Viscosity	1.802 x 10^{-5} kg/ms
Gravity	9.81 m/s^2
Lorry Height	0.222 m
Lorry Width	0.123 m
Lorry Frontal Area	0.0273 m^2
Free Stream Air Velocity	Taken from wind tunnel reading

Calibration of Data

To account for systematic errors or biases in measurements, raw data must be adjusted or corrected, a process known as data calibration. Data obtained from a variety of sources, such as sensors, instruments, or measurement equipment, must be accurate and reliable, which is why calibration is so important. Establishing a correlation between the measured data and the true or reference values is calibration's main objective. It assists in correcting for any inherent biases, errors, or fluctuations in the measuring system, ensuring that the obtained data is as accurate as feasible given the measurements.

Wind Tunnel Wind Speed Calibration

Data calibration for wind tunnel wind speed is an important process for ensuring the accuracy and consistency of wind speed data. By following the appropriate methods, data calibration for wind tunnel wind speed can help to improve the quality of data and the decisions that are made based on that data.

Weight Scale Calibration

Data calibration of weight scale is the process of ensuring that the data collected by a weighing scale is accurate and consistent. This is done by comparing the data to a known standard or reference value. Data calibration of weight is an important process for ensuring the accuracy and consistency of weight data. By following the appropriate methods, data calibration of weight can help to improve the quality of data and the decisions that are made based on that data.

RESULTS AND DISCUSSIONS

Numerical results were integrated with theories of aerodynamics and adequate justification. The aerodynamic forces affecting the model are then thoroughly covered in this chapter. The experiment was carried out by shifting the cylinder's position and flow visualisation by smoke wind tunnel testing. Microsoft Excel software was used to calculate and create the charts and tables that will be displayed, as well as to process and analyse the data. Since it can be used to evaluate the aerodynamic efficiency of various objects or vehicles, the coefficient of drag was computed for future study. The Reynold's Number and coefficient of drag were calculated using the raw data gathered from the simulations, such as the air velocity (m/s) and drag force. The Equation (2) and (3) below were used to obtain the Reynold's Number and drag coefficient, respectively:

$$Re_{height} = \frac{\rho_{ai}\ v_{\infty} L}{\mu} \qquad (2)$$

$$C_d = \frac{2F_d}{\rho_{ai}\ v_{\infty}^2 A} \qquad (3)$$

Where ρ is the density of the fluid which is constant (1.126 – 1.151 kg/m^3), v is free stream air velocity, L is characteristic linear dimension which is the height of the container and trailer (0.222 m), μ is the dynamic viscosity of the fluid (1.802 x 10^{-5} kg/ms), $F\ d$ is the drag force and A is the reference area (0.222 m x 0.123 m = 0.0273 m^2)

Wind Tunnel Testing

Based on Figure 6, it is observed that the majority of configurations that have the presence of fairing have positively affected the drag losses that are acting on the lorry. It is shown that the configuration Lorry with Fairing is the most effective configuration in reducing Cd. The percentage difference can be calculated using Equation (4) shown below for each case in relation to the initial drag coefficient. The percentage difference in drag coefficient has been tabulated in Table 2.

$$\%\ Difference = \frac{(Final - Initial)}{Initial} \times 100 \qquad (4)$$

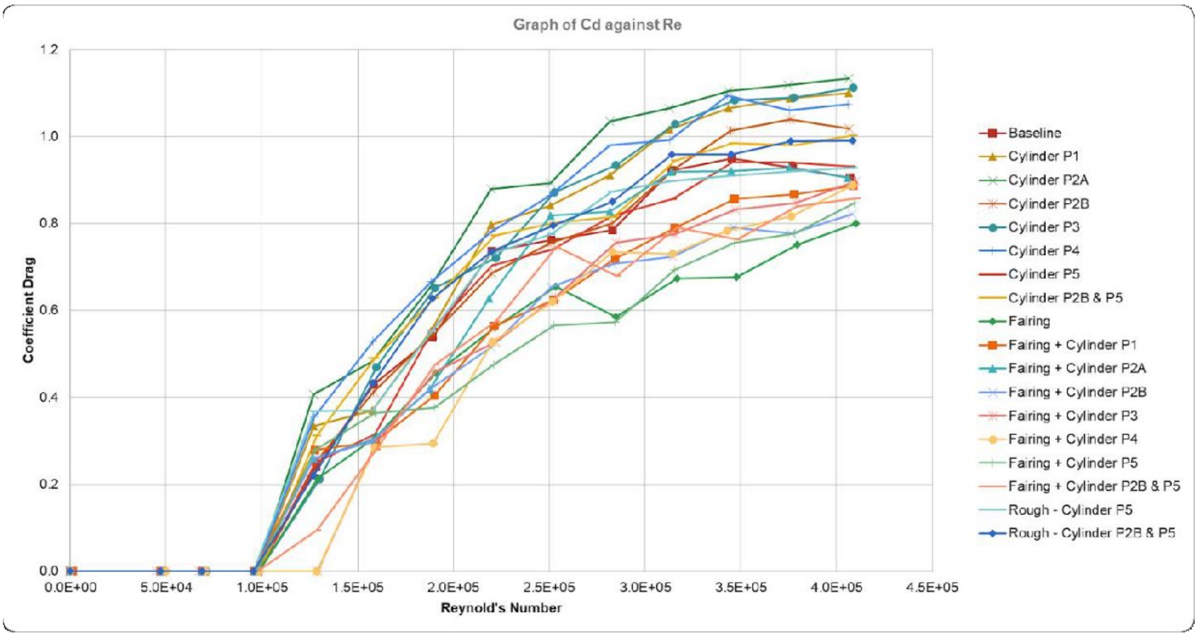

Figure 6: Combined graph of Cd against Re for all 18 tests [21]

Based on Figure 6 and Table 2, it is shown that all 6 configurations with the presence of cylinders (Lorry with Cylinder Positions 1, 2A, 2B, 3, 4 and 5) have negatively affected the drag losses acting on the lorry. Comparing the current data with the previous two papers, Kamid [19] and Hamdan [20], it is presented that all Cylinder Positions 1, 2A, 2B, 3, 4 and 5 have positively affected in reducing Cd. This could be due to the fact that, firstly, the lorry model used in the previous papers was simple and basic, whereas the current lorry model resembles a real-life lorry in Malaysia where it has more complex aerodynamic parts. This could affect the drag coefficient of the lorry. Secondly, there is a difference in the cylinder size ratio between the previous and current lorry model. Equation (5) shows the calculation to calculate the cylinder ratio.

$$Cylinder\ ratio\colon \frac{Cylinder\ diameter}{Height\ of\ container\ of\ lorry} \quad (5)$$

Table 2: Comparison of Cd for each cylinder configuration [21]

Configuration	Drag Coefficient (C_d)	% Difference
Baseline	0.904	-
Cylinder Position 1	1.100	21.68
Cylinder Position 2A	1.134	25.44
Cylinder Position 2B	1.018	12.61
Cylinder Position 3	1.112	23.01
Cylinder Position 4	1.074	18.81
Cylinder Position 5	0.931	2.99
Rough Surface Cylinder Position 5	0.928	2.65
Cylinder Positions 2B and 5	1.002	10.84
Rough Surface Cylinder Positions 2B and 5	0.990	9.51
Fairing only	0.880	- 11.50
Fairing with Cylinder Position 1	0.887	- 1.88
Fairing with Cylinder Position 2A	0.905	0.11
Fairing with Cylinder Position 2B	0.821	- 9.18
Fairing with Cylinder Position 3	0.898	- 0.66
Fairing with Cylinder Position 4	0.888	- 1.77
Fairing with Cylinder Position 5	0.847	- 6.31
Fairing with Cylinder Positions 2B and 5	0.858	- 5.09

Since both lorry models have different dimensions and sizes, the formula above is used in order to produce similar model performance. The previous and current lorry have a ratio of 0.125 and 0.133 respectively. This shows that the current model has a bigger cylinder than the calculated one. This is because there is an error in calculating the cylinder ratio where the length of the container of lorry is applied instead of the lorry height. The size of the cylinder could also affect the drag coefficient of the lorry. Thirdly and the most important reason, there is an error regarding the data taken for Baseline condition. Based on Figure 6, the trendline showed a minor decrement trend from Reynold's Number 3 x 10^5 to 4 x 10^5. The trendline illustrated is due to the lift of the lorry during testing starting at Reynold's Number 3 x 10^5. The lifted lorry causes the weighing scale to measure incorrect data due to the thread connected between the scale and lorry is not perpendicular anymore. By theory, the lorry does not have enough force to hold against the wind speed and hence produce higher drag. In this case, the baseline condition could have the highest Cd among all 18 configurations.

Figure 7: Lifting of Lorry (Baseline Condition) during testing [21]

On the other hand, based on the previous two papers, Kamid [19] and Hamdan [20], both findings prove that Cylinder Position 2B and Position 5 are the best configurations to reduce Cd in their respective research. Besides that, Cylinder Positions 2B and 5 produce a high Cd hence proving that applying two cylinders increases the drag of the lorry. Lastly, rough surface cylinders produce a lower Cd than the original cylinders. With reference to Table 2, configuration Rough Surface Cylinder Position 5 has a lower Cd percentage difference of 2.65% than configuration Cylinder Position 5 of 2.99%.

Figure 8: Lorry with brass placed [21]

Table 3: Comparison of Cd for each cylinder configuration in second experiment [21]

Configuration	Drag Coefficient (C_d)	% Difference
Baseline	1.016	-
Cylinder Position 5	0.990	- 2.56
Rough Surface Cylinder Position 5	0.988	- 2.76

Based on Figure 9 and Table 3, it is investigated that the Baseline configuration has the highest drag produced which is 1.016. On the other hand, the Rough Surface Cylinder Position 5 produced the lowest drag which is 0.988. However, at Reynold's Number 3.15 x 105, which converts to 22.5 m/s or 81 km/h, the Cylinder Position 5 produced the lowest drag with 0.745, followed with 0.859 and 0.886 by Rough Surface and Baseline configuration respectively. This illustrates that the smooth surface reduces more drag on laminar flow whereas the rough surface acts better on reducing drag on turbulent flow.

Smoke Wind Tunnel Testing for Flow Visualisation

To support the data produced from the experimental investigation, a smoke wind tunnel testing is held for flow visualisation of air acting up on the lorry. Four variants are chosen in this testing which are Lorry baseline, Lorry with Cylinder Position 2B, Lorry with Cylinder Positions 2B and 5, and lastly, Lorry with Fairing and Cylinder Position 2B.

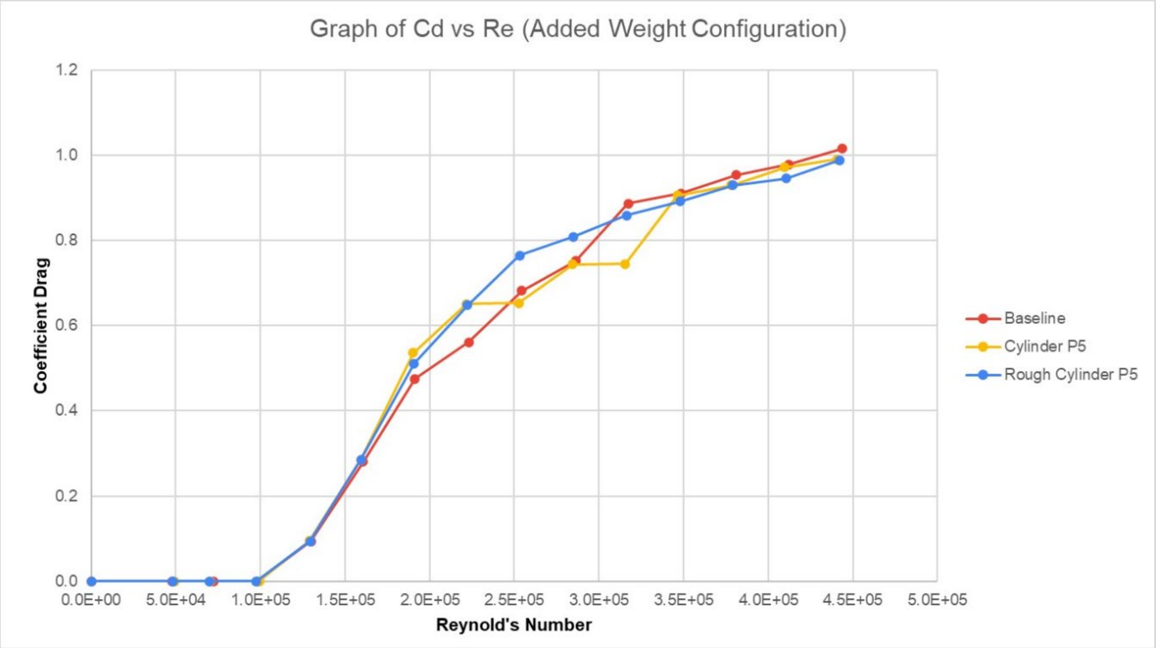

Figure 9: Combined graph of Cd against Re for the second experiment [21]

Figure 10: Lorry baseline during testing [21]

Photographs of the smoke flow visualisation were made with the lorry model to define its behaviour as a function when moving. Figure 10 shows the flow pattern of smoke of configuration Lorry Baseline. Flow approaching the roof stagnation point of the lorry is deflected upwards. Then, the flow approaching the top-front edge of the lorry container is deflected upwards, and the streamlines pass over the top surface of the lorry container. These deflected areas cause vortex formation, which in return increases the drag of the lorry model [22].

Figure 11: Lorry with Cylinder Position 2B during testing [21]

Figure 11 illustrates the flow pattern of smoke of configuration Lorry with Cylinder Position 2B. Flow approaching the roof stagnation point of the lorry is deflected upwards. Then, the flow approaching the top-front edge of the lorry container is deflected upwards, and the streamlines pass over the top surface of the lorry container, similar to the baseline lorry. These deflected areas cause vortex formation, which in return increases the drag of the lorry model. [22] However, the flow approaching the top-end surface of the lorry container passes quite smoothly towards the cylinder. The cylinder disrupts and disperses the air around it. It diminishes the turbulence that the moving vehicle creates. As a result, it assists in reducing air drag and turbulent flow. [19]

Figure 12: Lorry with Cylinder Positions 2B and 5 during testing [21]

Figure 12 emphasises the flow pattern of smoke of configuration Lorry with Cylinder Positions 2B and 5. Flow approaching the roof stagnation point of the lorry is deflected upwards. Then, the flow approaching the top-front edge of the lorry container is slightly deflected upwards, and the streamlines pass over the top surface of the lorry container, similar to the baseline lorry. These deflected areas cause vortex formation, which in return increases the drag of the lorry model [22], but the presence of Cylinder Position 5 reduces the vortex shedding at the particular area. Then, the flow approaching the top-end surface of the lorry container passes smoothly towards the cylinder. However, placing two cylinders on a lorry will increase drag. This is because the cylinders will disrupt the airflow around the lorry, causing the air to flow more turbulently. Turbulent airflow is less efficient than laminar airflow, which means that it will create more drag. In addition, the cylinders will also create vortices, which are swirling currents of air. Vortices can also increase drag, as they can create areas of high and low pressure that can disrupt the airflow around the lorry.

Figure 13: Lorry with Fairing and Cylinder Position 2B during testing [21]

Figure 13 elucidates the flow pattern of smoke of configuration Lorry with Fairing and Cylinder Position 2B. Flow approaching the stagnation point of the roof fairing passes smoothly. Then, the flow approaching the top-front edge of the lorry container also passes smoothly, and the streamlines pass over the top surface of the lorry container. The smooth flow of smoke is due to the presence of the roof fairing. A roof fairing reduces the aerodynamic drag on the vehicle by directing the onward wind flow smoothly onto the container and thus reducing flow separation in front of the container. [23] A highly curved and bulbous fairing helps reduce drag better, especially in the presence of side winds, similar to the current fairing. After that, the flow approaching the top-end surface of the lorry container passes smoothly towards the cylinder. The cylinder disrupts and disperses the air around it. It diminishes the turbulence that the moving vehicle creates. As a result, it assists in reducing air drag and turbulent flow better than other positions [19].

CONCLUSION

The main purpose of this research is to study the effects of static cylinders and their positioning as a drag reducer for container lorries. Configuration Lorry with Fairing is the most effective configuration in reducing Cd with a percentage difference of 11.5% compared to the lorry baseline. There is a setback in this project where all 6 configurations with the presence of cylinders (Lorry with Cylinder Positions 1, 2A, 2B, 3, 4 and 5) have negatively affected the drag losses acting on the lorry which could be due to several reasons. On the other hand, based on the previous two papers, Kamid [19] and Hamdan [20], both findings prove that Cylinder Position 2B and Position 5 are the best configurations to reduce Cd in their respective research. This demonstrates the importance of cylinder positioning, where a simple change in the vertical axis can increase or decrease the drag of the lorry.

Then, a second experiment was made to improvise the data where the configurations involved were the Baseline and the best cylinder configuration, which was Cylinder Position 5. In this experiment, the Rough Surface Position 5 was also tested to study the effect of drag on different cylinder surfaces. In order to prevent the lorry from lifting during wind tunnel testing, a 2kg weighed brass was placed on top of the lorry securely. Results show that Cylinder Position 5 reduces the most drag at lower wind speed and the Rough Surface acts better at higher speed. For smoke wind tunnel testing, the Fairing with Cylinder Position 2B condition has the best smoke flow as the streamline flows smoothly from the fairing and disperse minimally at the cylinder. It can be concluded that all the objectives have been met in this project.

In this research, there are many aspects that can be improved. First and foremost, the research can be further enhanced by upgrading the design of the cylinder. The cylinder is created as a simple cylinder for these tests. The experiments can also be performed in the future using a range of cylinder designs to explore the implications of the shape on the performance of the truck, where the shape is not limited to simply a standard cylinder. For instance, according to the literature, using a different cylinder design with higher roughness, such as porous cylinders or dimpled cylinders, can result in better drag reduction because these designs delay the formation of vortex on the spinning cylinder, which will in turn produce less drag. Other than that, proposing a rotating cylinder to the lorry could also reduce drag since the current lorry model resembles an actual lorry model in Malaysia.

In present study, induction heating technique is proposed to be effective method for SMA material compared to electrical heating technique. It has been found that the actuation gained by induction heating is good enough to use in practical applications. Induction heating is capable to actuate Flexinol wire once the optimum activation temperature is achieved, depending on the desired application. This study could provide the designer to accurately predict the coupling behaviour of SMA using induction heating.

ACKNOWLEDGEMENT

The authors would like to thank Associate Professor Dr. Azmin Shakrine Mohd Rafie for his invaluable guidance and support throughout this project. His enthusiasm and expertise, particularly in Aerodynamics, were instrumental in shaping this research.

REFERENCES

[1] Khalighi, B., Balkanyi, S. and Bernal, L., 2013. Experimental investigation of aerodynamic flow over a bluff body in ground proximity with drag reduction devices. International Journal of Aerodynamics, 3, pp.217-233.
[2] Solmaz, H. and İçingür, Y., 2015. Drag coefficient determination of a bus model using Reynolds number independence. International Journal of Automotive Engineering and Technologies, 4(3), p.146.
[3] Cihan, B., 2017. The experimental investigation of the effects of spoiler design on aerodynamic drag coefficient on truck trailer combinations. International Journal of Automotive Engineering and Technologies, pp.11-18.
[4] Schoon, R.E., 2007. On-road evaluation of devices to reduce heavy truck aerodynamic drag. SAE Publication, SAE Paper No. 2007-01-4294.
[5] Harun, C., 2013. A study on aerodynamic drag of a semi-trailer truck. Procedia Engineering, 56, pp.201-205.
[6] Snyder, R.H., 1997. Tire rolling losses and fuel economy. SAE Special Publication, SAE Paper No. 74.
[7] Landman, D., Wood, R.M. and Seay, W.S., 2009. Understanding practical heavy truck drag reduction limits. SAE Publication, SAE Paper No. 2009-01-2890.
[8] Schoon, R.E., 2007. On-road evaluation of devices to reduce heavy truck aerodynamic drag. SAE Publication, SAE Paper No. 2007-01-4294.
[9] Bayindirli, C., 2016. The determination of aerodynamic drag coefficient of truck and trailer model by wind tunnel tests. International Journal of Automotive Engineering and Technologies, 5(2), pp.53-60.
[10] Christensen, J., Glaeser, K.P., Shelton, T., Moore, B. and Aarts, L., 2015. Innovation in truck technologies. OECD Publishing.
[11] Perzon, S. and Davidson, L., 2000. On transient modelling of the flow around vehicles using the Reynolds equation. International Conference on Applied Computational Fluid Dynamics (ACFD) Beijing China, pp.720-727.
[12] Sarı, M.F., 2007. The aerodynamic analysis of air resistance affecting the front form of light commercial vehicles and its effect on fuel consumption. Osmangazi University, Institute of Science and Technology, Master Thesis, Eskişehir, pp.28-54.
[13] Modi, V.J., Hill, S.St. and Yokomimizo, T., 1995. Drag reduction of trucks through boundary-layer control. Journal of Wind Engineering and Industrial Aerodynamics, 54/55, pp.583-594.
[14] Gillieron, P. and Kourta, A., 2010. Aerodynamic drag reduction by vertical splitter plates. Experiments in Fluids, 48, pp.1-16.
[15] Fourrie, G., Keirsbulck, L., Labraga, L. and Gillieron, P., 2011. Bluff-body drag reduction using a deflector. Experiments in Fluids, 50, pp.385-395.
[16] Cooper, K.R., 2004. Commercial vehicle aerodynamic drag reduction: historical perspectives as a guide. In: The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains. McCallen, R., Browand, F. and Ross, J. (eds). Springer, New York, pp.9-28.
[17] Sabah Publishing House Sdn Bhd., 2022. Expanding with the Actros. Daily Express.

[18] Su'ud, A.I.A., 2019. HSTD, Hap Seng commercial vehicle serah 10 Jenama Actros Kepada Uniglory. Utusan Borneo.

[19] Kamid, M.I., 2022. Computational analysis on bluff body to reduce the drag force by using rotating cylinders as drag reducers (bachelor's thesis). Universiti Putra Malaysia.

[20] Hamdan, A.S., 2023. Experimental investigation on static and spinning cylinder and its positioning as a drag reducer on truck's container (bachelor's thesis). Universiti Putra Malaysia.

[21] Rizalman, M.A.H., 2023. Experimental investigation on cylinders and its positioning as a drag reducer on container lorries (bachelor's thesis). Universiti Putra Malaysia.

[22] Kusaiynov, K., Tanasheva, N.K., Turgunov, M.M. and Alibekova, A.R., 2015. Analysis of aerodynamic characteristics of rotating porous cylinders. Technical Physics, 60(5), pp.656-659.

[23] Krishna, M. and Ram, C., 2011. Adjustable roof fairing for truck aerodynamics. SAE Technical Paper 2011-26-0106.