Journal of Applied Fluid Mechanics, Vol. 17, No. 12, pp. 2592-2607, 2024.
Available online at www.jafmonline.net, ISSN 1735-3572, EISSN 1735-3645.
https://doi.org/10.47176/jafm.17.12.2664

Optimizing Fuel Efficiency in Intercity Buses: Aerodynamic Design Enhancements and Implications for Sustainable Transportation in Bangladesh

M. R. Rony†, M. J. Islam, S. Shahriare and M. M. Alam

Department of Mechanical Engineering, Chittagong University of Engineering and Technology, Chattogram, Raozan-4349, Bangladesh

†*Corresponding Author Email: u1903063@student.cuet.ac.bd*

ABSTRACT

This study explored the aerodynamic aspects of the Hyundai Universe Express Noble bus, a common passenger bus model in Bangladesh, and proposed modifications to improve its performance. The airflow around the bus was analyzed using Computational Fluid Dynamics (CFD) simulations. Consequently, areas of high drag and turbulence were identified. These results led to the redesign and testing of several shape modifications for the bus, including adjustments to the roof spoiler, side mirrors, and front grille. After finding the bus model with the lowest drag coefficient, that model is further analyzed to find the aerodynamic effects of side windows on a bus and their impact on fuel efficiency. The aim is to determine how much side windows significantly affect fuel efficiency. The aerodynamic effect with windows open and closed is also evaluated after identifying an appropriate model. Model 1.2 uses 7.72% less fuel than base Model 1.0. Model 1.2 with windows uses 2.5% more fuel than Model 1.2. The study also evaluates the fuel cost per 500 km for all models suggesting that non-ac buses with side windows consume slightly more fuel than AC buses. Changing the bus shape to Model 1.2 will reduce the drag coefficient by 8.67% and fuel consumption by 7.72%. The study offers insights into reducing drag force, minimizing air resistance, enhancing exterior styling, and improving vehicle stability. Furthermore, these findings have practical implications for the transportation industry as they demonstrate the potential for improving the efficiency and sustainability of large vehicles through aerodynamic design.

Article History

Received February 18, 2024
Revised May 26, 2024
Accepted June 8, 2024
Available online October 2, 2024

Keywords:

Aerodynamics
Bus
Computational Fluid Dynamics
Reynolds-Averaged-Navier-Stokes
Side windows

1. INTRODUCTION

The modern generation relies heavily upon vehicles for a faster communication system. However, there have been numerous fuel crises due to the massive population growth and the demand for automobiles. Hence the world's most pressing issue in the upcoming decades will be the acceleration of fossil fuel decomposition. The two most essential petroleum products, diesel, and gasoline, degrade daily, consisting of 74% of crude oil (Wisconsin K-12 Energy Education Program 2020). Mobility constraints in 2020 significantly influenced the transport industry, accounting for around 60% of the overall oil consumption (Energy Agency, 2021). So, figuring out new ways to lower our fuel consumption is therefore crucial to maintaining the efficacy of the vehicle's monitoring system. For the past 15 years, Bangladeshi roads have undergone tremendous improvement, and travel times have decreased because of their increased speed (Government of the People's Republic of Bangladesh, Ministry of Communications, Roads and Railways Division, 1999; Government of The People's Republic of Bangladesh, Ministry of Road Transport and Bridges, Road Transport and Highways Division, et al., 2016). Aerodynamics affects several aspects of a car, including quality, directional stability, wind noise stability, cleaning of lights, windows, and body, cooling of an engine, gearbox, and brakes, and, finally, passenger compartment heating, ventilation, and air conditioning (Pfeiffer & Wriggers, n.d.). The tractive resistance, which results from both rolling and aerodynamic resistance, is overcome by the engine power in a moving vehicle. Rolling resistance, however, is overpowered when the vehicle's speed rises due to aerodynamic resistance. The engine is under less strain when there is less aerodynamic resistance at high speeds, which increases the fuel efficiency of the bus (Hucho & Sovran, 1993). The drag force is the result of both pressure drag and viscous drag. Viscous drag is the force brought about by the fluid's shear stresses acting on the item's surface. Pressure drag is caused by the net differences between the object's front and back ends due to wake regions. Pressure drag is the leading force in the majority of applications, such as moving buses, and streamliners are employed to lessen this force. There is no pressure drag on a fully streamlined piece (only viscous drag). Solid structures are often designed to be more streamlined to lessen the pressure drag. A simplified body allows air to gradually slow down along the rear of it, preventing boundary layers from separating. Separated boundary layers increase pressure drag and should be stopped since these forces would cause a vehicle to experience the same amount of drag (Akmal Bin Nasrul Hisham, 2008).

Vehicle stability is indirectly improved by reducing drag. The vehicle can accelerate to higher speeds more quickly by lowering drag. This can improve its handling and responsiveness, which increases overall stability, particularly when lane changes or cornering are involved. In general, vehicles with lower drag coefficients are less vulnerable to gusts and crosswinds. Stability is improved by this increased resistance to wind disturbances, particularly at faster speeds or in severe conditions. Improving the overall body shape of the vehicle also helps with handling and stability. In most nations, buses are a vital form of public transportation. Open-window buses without air conditioning are a common form of urban and intercity transportation. The combination of humid and warm weather and a high occupancy rate makes riding in these buses unpleasant

(Nikam & Borse, 2014). Given that buses use a lot of fuel, improving the aerodynamics of their designs will help reduce the amount of fuel that the vehicles use. Aerodynamics is a crucial aspect of vehicle design since it affects fuel efficiency and resistance to force, two factors that further make the vehicle necessary for human usage.

To be more precise, research on automobiles focuses primarily on decreasing their drag coefficient. Lower resistance to forward motion permits lower power output at the same time higher speeds for the same power output. Aerodynamics is the tool used to redesign a body shape that minimizes the opposing force obstructing the forward motion and drag forces while maximizing downforce and negative lifts (Damissie & Ramesh Babu, n.d.). The bus's aerodynamically efficient design lowers drag force, increasing fuel efficiency. So, the primary objective of this research is to

❖Redesign an intercity bus that has less aerodynamic resistance in its design.

❖Analyze the model using ANSYS FLUENT to minimize the drag force and drag coefficient.

❖Compare several models and find the effect of side windows (non-AC) following the best model selection, implementing the results obtained from the simulation study.

In this study, the aerodynamic effectiveness of a bus is compared to a typical bus design. After the identification of additional components that can reduce drag and increase effectiveness, this experiment will involve the building of a bus model as well as the measurement of the drag coefficient. The drag coefficient can be decreased to lessen drag. Calculating the fuel consumption rate of various is also carried out using this simulation. This study examines the airflow inside a bus to create affordable, eco-friendly ways to improve passenger comfort and decrease drag. A bus with open windows is analyzed here for airflow, and the results of drag force, drag coefficient (C_d), and fuel efficiency are contrasted with a bus consisting of closed windows as demonstrated in Fig. 2.

The study employs ANSYS Workbench 2021 and Fluent 2021 tools, for CFD (Computational Fluid Dynamics) analysis to forecast physical parameters on a bus's exterior, as well as pressure, velocity distribution, and analysis of drag forces. The initial base model is the Hyundai Universe Nobel demonstrated in Fig.1 It is a reasonably standard vehicle in Bangladesh's transportation industry. Following the initial simulation of the base model, three modifications are made to this model. The three designs are named as models 1.1, 1.2, and 1.3. Making the models more streamlined is the general strategy for modification. The drag force is quite strong and the stability is also not up to the mark at higher speeds because the base bus model has a flat front, which has increased air resistance. Hence the modification of the outer body design is done using the current model as a guide to solve these issues.

Upon completion of the simulations, the best model is chosen by comparing them, and again, windows are added to that model. Using a practicable k-ε turbulence model, the RANS (Reynolds-averaged Navier-Stokes) equations are solved to determine the flow distribution. The best model is further analyzed to find the aerodynamic effect of the side windows of the bus. New drag force and coefficient are measured by incorporating windows on both sides of the vehicle. These findings suggest optimum design on how windows affect fuel efficiency by calculating how much fuel would be saved per year theoretically with the most aerodynamic model compared to the base model. It will help realize the difference between AC and non-AC bus fares if the chassis remains the same in both cases.

2. LITERATURE REVIEW

The drag coefficient is a key determinant of a vehicle's aerodynamic performance, directly impacting engine requirements and fuel consumption (Vinayagam et al., 2017). In general, even a small reduction in drag can result in increased fuel economy.

Wind tunnel experiments and computational testing are two approaches that have been performed to show the efficacy of the novel concept design. According to the test results conducted by (Muthuvel et al., 2013), there has been an obvious 30–34 % decrease in drag force between the current bus and the new idea, and every 100

Fig. 1 Isometric view of the base Model 1.0

kilometers, 6–7 liters of fuel are used. (Damissie & Ramesh Babu, n.d.) redesigned the Yutong cross-country bus model and obtained a reduced drag coefficient which in turn resulted in less fuel consumption. Based on a review, the second alteration yields the smallest drag coefficient (0.535). For the new shape, the decrease percentage is 38.86%. The design was created and tested in a wind tunnel 6.79% difference once it had been finished and the drag coefficient had been determined. The experiment was done by (Akmal Bin Nasrul Hisham, 2008) acquiring an outcome drag coefficient of 0.574, which indicates a (Jadhav & Chorage, 2020) conducted a study where they modified the Volvo B11 bus and compared the drag coefficient to determine the best aerodynamic bus model. The bus models were analyzed with a CFD simulation where inlet air velocity ranged from 80-100 km/h. The high-velocity range was since aerodynamic characteristics are prone to a higher velocity range.Belachew et al. (2021) studied to create a framework for a simplified frontal body geometry and aerodynamic shape optimization for the FSR Isuzu bus model using available CFD software. For use as intercity buses in Ethiopia, (Tefera et al., 2023) modified Isuzu buses to have an aerodynamically efficient rooftop luggage compartment. It was discovered that the updated bus has an improved ergonomic design and uses less fuel. Alexei Pichardo-Orta et al. (2022) examined the airflow through an urban bus model's open windows or natural ventilation, and the resulting aerosol emissions that disperse throughout the passenger compartment. The findings of this work's numerical models and experiments all pointed to a significant increase in aerosol expulsion in this novel configuration and a reduction in the average air duration to 50 seconds by integrating frontal air intake into the bus. Using computational fluid dynamics analysis based on steady-state Reynolds-averaged Navier-Stokes turbulence models, (Al-Saadi et al., 2022) suggested aerodynamically optimized sport utility vehicle exterior designs. The goal of their study was to modify the front and upper sections of this particular car model. To improve streamlining, the spare tire on the back door and the rear diffuser are utilized as aerodynamic devices in conjunction. The findings of Yudianto et al. (2021) showed that during the platoon, a crosswind erodes aerodynamic advantages. The inter-bus distance controls the behavior of the airflow surrounding the bus, resulting in the differences in aerodynamic advantages that buses experience. These easy, reasonably priced aerodynamic drag reduction devices have been designed for use on a tractor-trailer truck. These devices accumulated over 85,000 kilometers of use throughout rigorous operational testing. By combining the technologies, 10% less fuel was used at an average speed of 47.5 MPH. Wood and Bauer (2003) demonstrated that with an appropriate drag coefficient of 0.45, this increase in fuel efficiency corresponds to an equal drag reduction of about 30%. In the investigation of Shekar et al. (2014), the paper examined how the vehicle's overall aerodynamic drag is affected by this "transverse outer profile.". The transverse outer profile creates skin friction, vortices, and drag, which can be understood using computational fluid dynamics (CFD). Consequently, methods for improving the transverse outer profile to reduce this drag component are assessed. Abinesh and Arunkumar (2014) experimented with two model bus bodies that were used in another study to lower the drag force. These are specifically models 1 and 2. Type 2 is the adaptation of the modified Volvo intercity bus, whereas model 1 is the current Volvo intercity bus type. A total of 10% less aerodynamic drag force was produced with the modifications during the study. The study of (Garcia-Ribeiro et al., 2023) on drag reduction of commercial buses showed that The CFD results and the experimental ones were in good agreement; where the largest drag coefficient reductions were 7.72% and 8.663%, respectively. These results therefore imply that significant aerodynamic improvements can be made to currently in-use commercial buses. (Raveendran et al., 2009) indicated that redesigning the exterior from the perspective of aerodynamics and aesthetics reduced a substantial amount of drag force and increased passenger comfort. Mukut and Abedin (2019) claim that passive flow control techniques alter the vehicle body's shape and control the flow field to achieve the desired impact on drag reduction. In another study, the passive flow control strategy improved the drag force of the 1/33 size bus model. Here (Bayindirli, 2019) used a bus model and found the impact of passive air canal application on drag coefficient by examining both numerically and experimentally. According to this study, fuel consumption can be reduced by roughly 2-6% at high vehicle speeds due to the aerodynamic gains made possible by using the passive flow management method. In a study conducted by Alonso-Estébanez et al. (2017), the study demonstrated the value of CFD in enhancing traffic safety, examining the functionality of the articulating wind fence, and identifying the box deck geometry parameters that significantly affect bus stability. Mezarciöz et al. (2010) in their investigation found that a comparison between the experimental results of Particle Image Velocimetry (PIV) and the predicted results for time-averaged flow data, specifically around the forward face of the model, revealed that the experimental results and the numerically predicted present results agreed well. Another study was attempted by Amen Mohamed et al. (2015) to lower the aerodynamic drag to lower the gas emissions from buses. Investigations were done on six distinct cases. A computational model was created to carry out this investigation. It was shown that aerodynamic drag can be reduced by up to 14%, resulting in an 8.4% decrease in fuel consumption. Additionally, the investigation showed that in various scenarios, the bus's aerodynamic drag can be predicted using the Neuro-Fuzzy approach. A three-dimensional model of the bus was run at 83 km/h with a Reynolds Number of 6.4 million at a zero-degree yaw angle. In comparison to other passenger buses of comparable dimensions, the simulation's drag coefficient of 0.25 is extremely low. When operating at 83 km/h, this model is verified to have 40% less air resistance than the buses that are currently on the market and are the same size. According to the results of Thomas et al. (2006).

CFD analysis and wind tunnel simulations, there can be a 2% or 5% increase in drag for single apertures and numerous openings respectively. In a study by Thorat et al. (2011) they modified the interior design of the vehicle to accommodate commuters' needs. At a speed of 100 km/h, the redesigned exterior body results demonstrated a reduction in C_d from 0.581 to 0.41 and an overall reduction in aerodynamic drag of roughly 30% as a result of the combined effects of reduced C_d and frontal area. The majority of passenger travels worldwide take place in open-window buses, where the comfort of motion-induced airflow. Such a bus's aerodynamics were investigated. Using a 1:25 model, flow visualizations in a water channel revealed extremely chaotic inflow through the front window and outflow through the rear window, respectively (Yelmule et al., 2009). Arteaga et al. (2020) performed research that aimed to investigate the aerodynamics of the bus body using CFD software and a wind tunnel. To validate the results obtained in the simulation software, various bodywork geometries were evaluated by printing 3D prototypes on a 1:200 scale. The results showed that the second option had an optimal drag coefficient and a higher lift coefficient with the reduction of turbulence zones and a body formed by two spoilers, followed by a body with the original body (Belachew et al., 2021).

Oforji et al. (2023) conducted a study to assess how the rhythm of window openings affected the patterns of passive airflow in low-rise residential buildings in Obosi, Nigeria. Temperature, relative humidity, and wind speed were measured for the study using the Anemometer TA465 device. The type, size, and intensity of the breeze were found to be significantly correlated with each other in the results. The study also discovered differences in indoor temperature, particularly between the lowest and highest floors. The results point to the necessity of using architectural rhythm design techniques to enhance low-rise building airflow patterns, particularly in densely populated urban areas. Gohari et al. (2022) investigated how to lower airborne noise by altering the side view mirror of a small sedan car. Utilizing both road testing and CFD simulation, the original mirror was evaluated. There was an 8–12% decrease in sound pressure with the updated VSVM geometry. This shows that the altered mirror may be able to lessen airborne noise, which could lower the possibility of annoying noise and improve passenger comfort while driving. The study emphasizes how crucial it is to make cars quieter. A natural ventilation system using air ducts positioned above the ceiling of the corridor is suggested by a study done by Ulum et al. (2023). The E-ITERA building, which has tiny windows and a glass facade, was selected as a case study. CFD software simulations demonstrated that when the WWR air duct and the WWR window are identical, cross-ventilation can occur and ventilation performance is enhanced. The outcomes demonstrated that cross-ventilation could be produced by the air duct. When the WWR air duct matches the WWR window, ventilation performance improves. The room's center had the greatest air velocity of 0.6 m/s.

Very few studies have been conducted to redesign a bus model from the perspective of the aerodynamic impact of open or closed windows. This investigation focuses on how large vehicles' windows and mechanical ventilation systems affected their aerodynamics. It mainly looks at a MAN city bus and the aerodynamic implications of having windows open and air conditioning systems within. The current research investigation focuses on the bus's aerodynamic characteristics, especially drag, which has a direct impact on fuel consumption. With the help of the commercial program ANSYS-Fluent, a computational model was created to forecast the aerodynamic performance of buses and the impact of windows integrated into the bus models.

3. VEHICLE DESIGN

To conduct the research, the first step in the methodology is to design the vehicle geometry. SolidWorks 2021 software is used to create 3D models of the buses. The aerodynamic design of a vehicle is meticulously engineered to optimize performance and efficiency, with each component playing a critical role. At the front, where airflow first encounters the vehicle, there's an area of overpressure and stagnation, influencing drag and pressure distribution. Rounding up the leading edges across the front is essential for boundary layer separation on the sidewalls and roof, minimizing aerodynamic drag. Moving towards the rear,

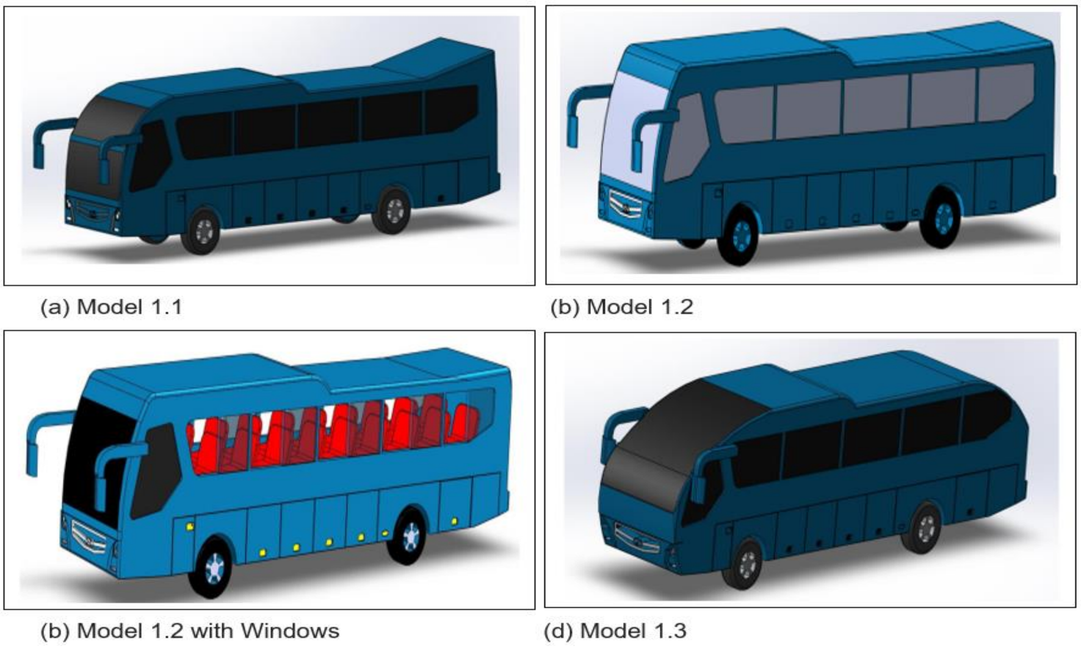


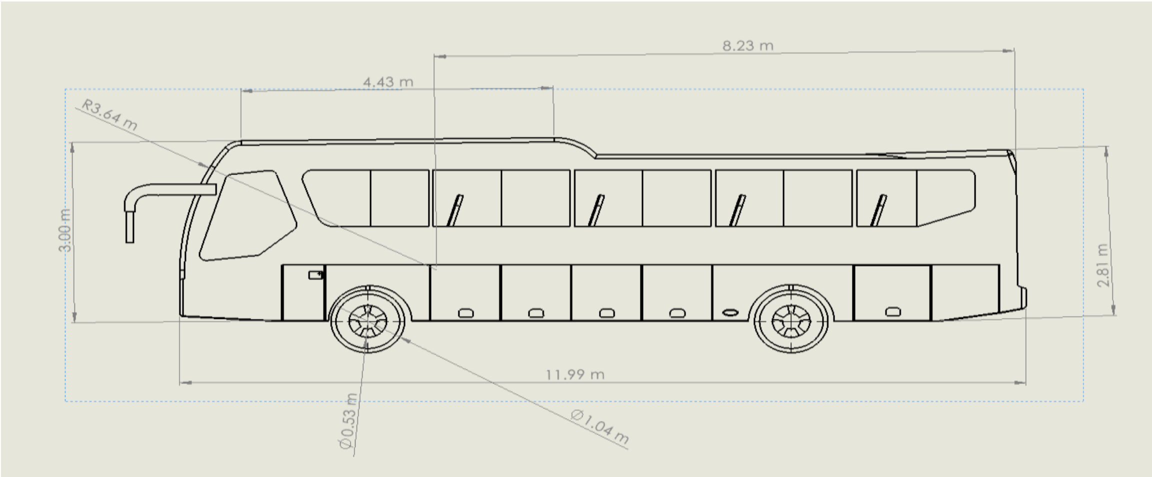
Fig. 2 Different configurations of the bus models

turbulence intensifies, particularly at the rear wall, where heavy mixing occurs, creating a continuous pressure differential below atmospheric pressure. Meanwhile, the underbody gap, surrounded by uneven, moving surfaces, serves as a crucial area where airflow streams downstream and outward, contributing to overall aerodynamic stability and efficiency. Each aspect of the vehicle's aerodynamic profile is finely tuned to optimize performance, reduce drag, and enhance fuel efficiency. The bus model was initially designed based on the standard bus shapes in Bangladesh's transportation industry. The bus currently in use is the base model (Model 1.0) the Hyundai Universe Express Noble bus depicted in Fig 1. Model 1.1 is the first bus model modified to have an aerodynamic shape to which a high deck shape is given. Its windshield consists of two separate parts. A rear spoiler is implemented and the AC vent is at the front. The second is less tapered than Model 1.1 and is named Model 1.2. It does not possess any rear spoiler. The third modification is named Model 1.3. It is filleted in both front and back. It is designed considering giving a downward streamlined airflow path. Neither in front nor rear side is equipped with a spoiler in it. In all three reshaped buses, the AC vent is kept on the front side. Total bus length, width, and height were kept the same. However, due to tapering on both sides of Model 1.3, interior space is slightly reduced. All the models have approximately similar lengths, widths, and heights. Making the models more streamlined was our key strategy of modification. The models are designed with a scale of 1:1. Dimensions of the buses are demonstrated in Fig. 3. The isometric views of different bus models are displayed in Fig. 2. The base bus model has a flat type front which is the main reason for the frontal high-pressure zone. It is acting like a bluff body. Modification of the front side was a high priority for the proposed models. The box-shaped frontal area increases air resistance and drag, making it less streamlined. Higher ground clearance contributes to reduced aerodynamic performance. While these design choices prioritize passenger comfort and practicality, they compromise the bus's overall aerodynamic efficiency. Hence three Modifications to the base model were made. We've created three versions in this case, each with a unique exterior body design and appearance to maximize the impact of each adjustment on the drag force. Upon completion of the simulation, we chose the best model by comparing drag coefficients, and again, windows were added to that model. This windows-featured model is again simulated to determine the aerodynamic impact of windows on a bus. The study also provides a theoretical assumption as to how much fuel will be saved per 500 km with the most aerodynamically effective bus model compared to the other models. In the first shape Model 1.1 with a high deck shape, its windshield consists of two separate parts.

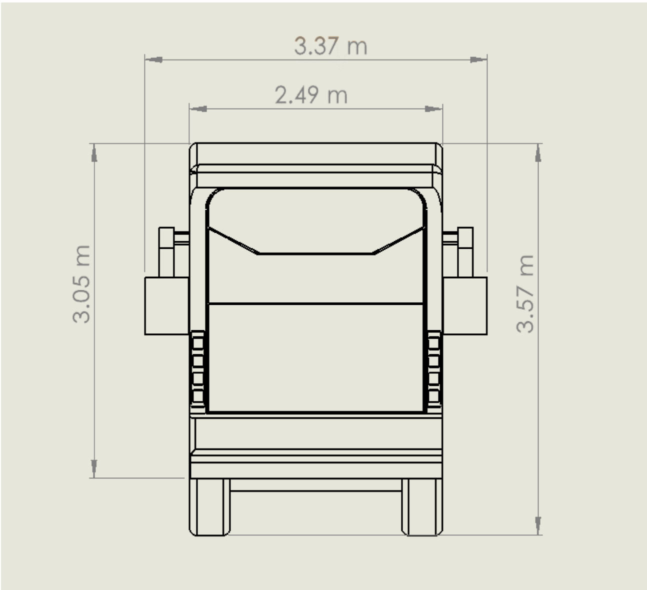
A rear spoiler is implemented and the AC vent is at the front. Model 1.2 has a single tangent front and it has no spoiler at the rear. It has the lowest drag coefficient and therefore windows have been implemented to it for further analysis. Model 1.2 with side windows, is the type of shape that is widely used in non-ac bus sectors. Model 1.3 is the third shape which has a more streamlined front and rear. This type of bus is also known as a bullet bus because it has a shape like a bullet. This model was assumed to have the lowest C_d but analysis shows this bus is the least aerodynamic.

3.1. Meshing

After designing the bus models the next step is to conduct the meshing process. ANSYS WORKBENCH preprocessor and FLUENT post-processor software tools were utilized to create the mesh and resolve fluid dynamics issues. Meshing is the process of dividing up a continuous geometric surface or domain into a grouping of more tiny, related components, or cells. This approach



(a) Side view



(b) Front view

Fig. 3 Dimensions of bus model configurations

is utilized in computer-based simulation methods for Finite Element Analysis (FEA) and Computational Fluid Dynamics (CFD) (Rashwan, 2019). It denotes the partitioning of a simple CAD model into small elements and specific node locations. In the case of CFD analysis to prevent numerical diffusion, FLUENT needs a high-quality mesh. The skewness is the main metric for mesh quality, though there are other metrics as well (Vinayagam et al., 2017). A smooth and high-quality mesh must be established for the numerical analysis of the finite volumes to produce accurate results. Thus, in numerical studies, mesh quality is greatly sought. Achieving the required mesh quality gets harder as the studied geometries get more complicated. The bus model has a few tiny, curved elements. In this instance, the required level of mesh quality could not be achieved. To attain the intended mesh quality, additional data drawing simplification, the creation of flat rather than arched parts, or the closure of small sections and the generation of more mesh structures are required. The drawing data diverges from the geometric similarity in this instance. The average element quality in this study is 0.68. It is not advised to have this value greater than 0.5 (Bayindirli, 2019).

Independence from the mesh in the numerical solution is achieved if, after a given value, the result remains unchanged despite an increase in the number of meshes. The mesh independence of the study was assessed considering the findings of (Bayindirli, 2019). In his study, 650306-6011160 mesh numbers were studied where the Reynolds number range was 382866-792900. During the grid study, the C_d values varied from 0.441-0.586, which is similar to the findings of the current study. This provides a mesh independence study for the current analysis Using unstructured grids, the computational domain was discretized. The flexibility of producing sufficient computational points in areas with severe gradients is typically guaranteed by these kinds of grids. Figure 4 shows the tetrahedral elements that covered the computational domain. Near the solid boundary, the grid is extremely fine. The dimensionless

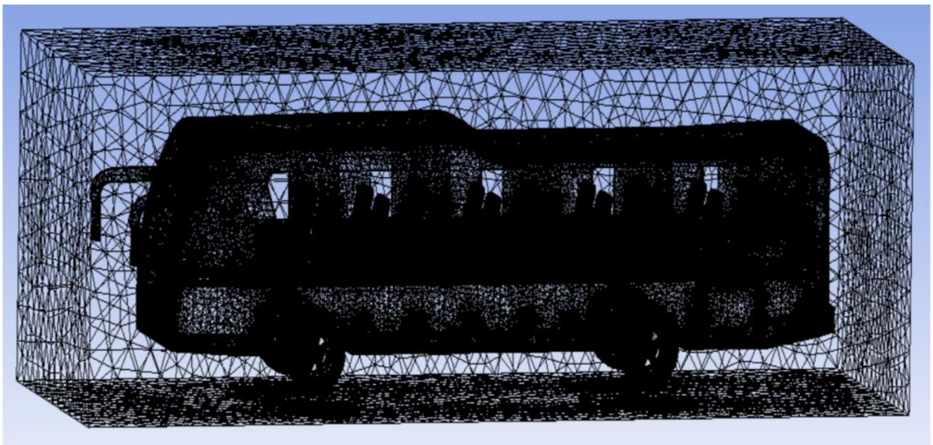


Fig. 4 The meshing of the bus model

distance between the wall and the first computational point i.e., y^+ is nearly about 1.8 (Amen Mohamed et al., 2014). This can be computed using the following formula:

$$y^+ = \frac{U_\tau y}{\upsilon} \quad (1)$$

$$U_\tau = \frac{\tau}{\rho} \quad (2)$$

where U_τ is the friction velocity, υ is the kinematic viscosity, and y is the distance to the first point off the wall. ρ is the flow density, and τ is the wall shear stress. A good resolution of the complex turbulent flow is ensured by the value of y^+.

Both the domain's surface and the vehicle's geometry were used to create the surface mesh. The computational grid was created between the domain and the vehicle's surface. To capture specific regions of interest (where separation may transpire and turbulence intensity is elevated), the cells must be sufficiently small to resolve all irregularities and produce a stable solution (Abinesh & Arunkumar, 2014). The number of nodes and elements of the bus models are shown in Fig.5.

In the case of the number of elements and node count, due to the complex turbulence of the fluid medium intercepted in the vehicle, Model 1.2 with open windows and Model 1.2 with windows (shown as 1.4 in Fig. 5) consists much bigger number compared to the other ones.

4. BOUNDARY CONDITIONS

ANSYS FLUENT 2021 was utilized to apply boundary conditions to the meshed model. The simulation took into account a 100 km/h straight wind condition. The ambient air temperature was considered 25°C. this surrounding air was taken as an incompressible fluid type with a density of 1.225 kg/m^3. An inlet condition of constant velocity was applied to resemble the conditions of constant wind velocity observed in wind tunnel tests. Operational pressure at the outlet was equivalent to gauge or atmospheric pressure. The specifics for all boundary conditions that

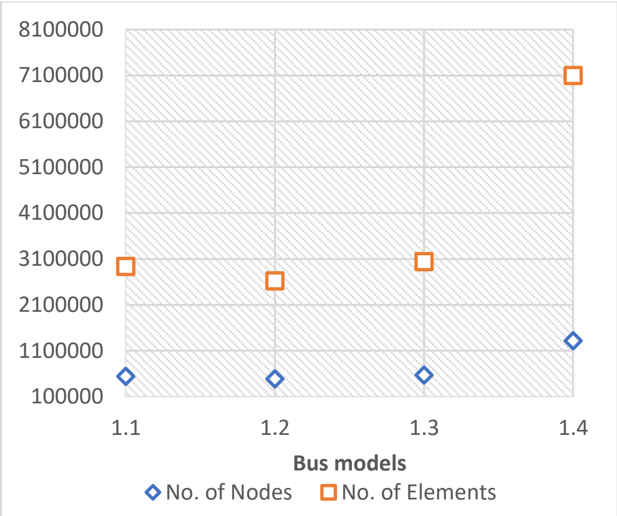


Fig. 5 No. of node and element count in the vehicle model specification

were applied during the analysis are included in Table 1 (Damissie & Ramesh Babu, n.d.).

Selecting an inlet velocity of 100 km/h is determined by a few factors. It indicates the typical speed for the buses under the study's operating circumstances. For instance, the buses normally travel on highways or other comparable roads at a speed of about 100 km/h, this velocity would be important to simulate their aerodynamic performance under typical operating conditions demonstrated in Fig. 6. The bus models can travel in less velocities. However, the best-accumulated values are obtained from the higher velocity conditions as evident from the grid independence study. Furthermore, it has been selected because it is within a range of speeds that are frequently seen in actual driving situations and aerodynamic effects are prone to high velocity. This ensures that the simulations accurately represent the aerodynamic effects and pertinent flow physics that the buses encounter in their regular operations.

The finite volume method is used to solve the governing equations of the current model's proposed

Table 1 Boundary conditions applied on the bus models

Boundary	Boundary Conditions	Values
Inlet	Constant velocity Turbulent-intensity Length-scale	V=100 km/h
Outlet	Pressure based outlet	Constant pressure =0 Pa
Vehicle body	No slip-condition Stationary wall	-
Road	Moving wall-No slip	V=100 km/h
Domain	Stationary wall with specified shear	Shear stress=0 Pa

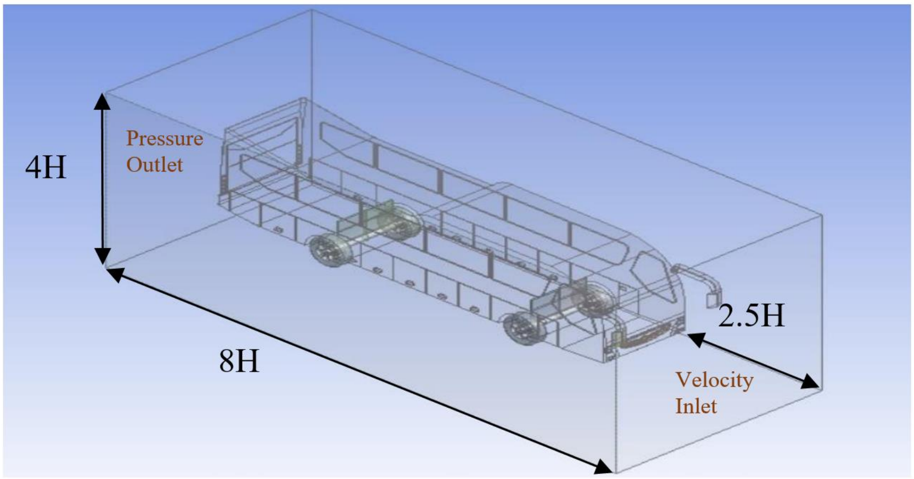
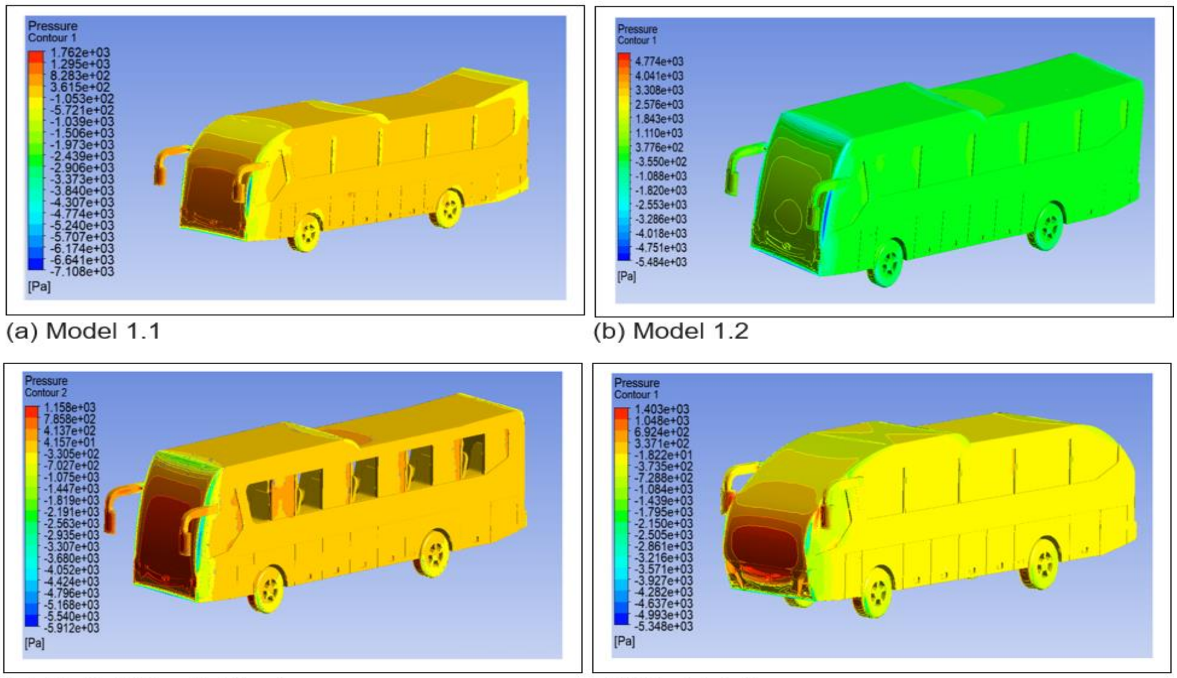


Fig. 6 ANSYS -modeled bus's computational domain and boundary condition



(a) Model 1.1 (b) Model 1.2

(c) Model 1.2 with Windows (d) Model 1.3

Fig. 7 Pressure contour of the vehicle specifications

domain. This method discretizes the problem into a certain number of control volumes. Equations for continuity and Navier-Stokes are also known as differential motion equations. Solving these equations provides the pressure and velocity components for the studied simulations. Using computational fluid dynamics (CFD), the finite volumes are solved using continuity and momentum equations. Analytically solving these equations is challenging in real life. Consequently, ANSYS FLUENT software is used to solve these equations numerically.

Continuity Equation

As the mass balance in the control volume of a flow, the continuity equation is expressed as

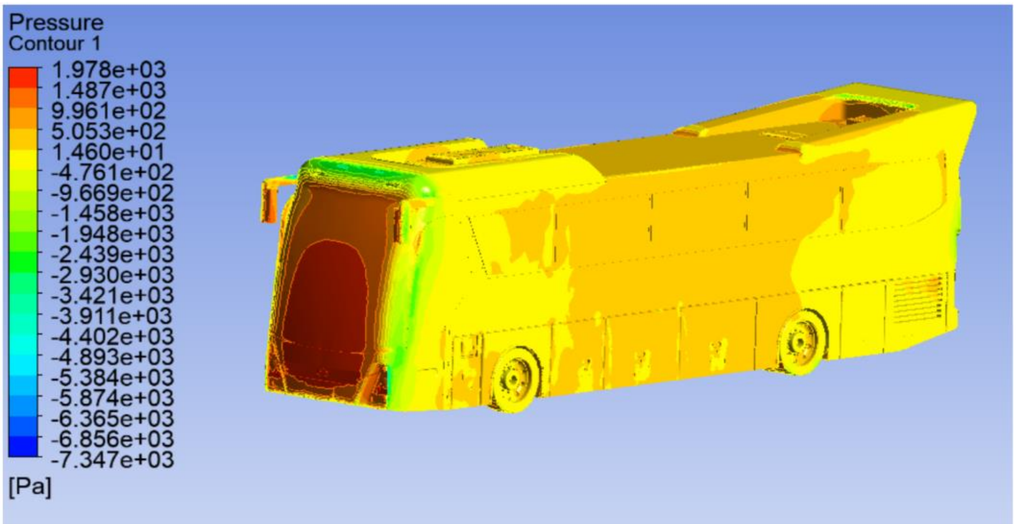


Fig. 8 Pressure contour of the base Model 1.0

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \quad (3)$$

Momentum Equation

Newton's second law states that the sum of the forces acting on a fluid fraction equals the rate of change of that fluid fraction's momentum. The unit volume of a fluid fraction's momentum increase rate in the x, y, and z directions is expressed in terms of $\rho\frac{Du}{Dt}$, $\rho\frac{Dv}{Dt}$, $\rho\frac{Dw}{Dt}$ (Ince, 2010) respectively.

The equation for momentum's x-component:

$$\rho\frac{Du}{Dt}=\frac{\partial(-P+\tau_{xx})}{\partial x}+\frac{\partial\tau_{yx}}{\partial y}+\frac{\partial\tau_{zx}}{\partial z}+S_{M_x} \quad (4)$$

The equation for momentum's y-component:

$$\rho\frac{Dv}{Dt}=\frac{\partial\tau_{xy}}{\partial x}+\frac{\partial(-P+\tau_{yy})}{\partial y}+\frac{\partial\tau_{zy}}{\partial z}+S_{M_y} \quad (5)$$

The equation for momentum's z-component:

$$\rho\frac{Dw}{Dt}=\frac{\partial\tau_{xz}}{\partial x}+\frac{\partial\tau_{yz}}{\partial y}+\frac{\partial(-P+\tau_{zz})}{\partial z}++S_{M_z} \quad (6)$$

Navier – Stokes Equations

Differential motion equations can also be called Navier-Stokes and continuity equations. Pressure and the three components of velocity (x, y, and z) are calculated after these equations are solved under certain assumptions (Ince, 2010). The most practical approach for developing the Navier-Stokes equations under finite volume method:

$$\rho\frac{Du}{Dt}=-\frac{\partial P}{\partial x}+div(\mu\ grad\ u)+S_{M_x} \quad (7)$$

$$\rho\frac{Dv}{Dt}=-\frac{\partial P}{\partial y}+div(\mu\ grad\ v)+S_{M_y} \quad (8)$$

$$\rho\frac{Dw}{Dt}=-\frac{\partial P}{\partial z}+div(\mu\ grad\ w)+S_{M_z} \quad (9)$$

5. RESULT AND DISCUSSION

This section includes the CFD analysis results for each case: static pressure distribution on the vehicle, drag coefficient, and velocity streamline flow sequentially.

5.1. Pressure Contours

Figure 7 depicts the pressure concentration of the bus models. The pressure concentration in the front area of the bus is visible in the static pressure distribution plot of the bus body at 100 km/h as the airflow strikes and momentarily rests there. The base model was distinguished by a flat-front design, which contributes substantially to the formation of a high-pressure zone at the frontal region of the vehicle, similar to a bluff body. In addition, the side mirrors were sharp, lacked smooth edges, and had a flat shape. The rooftop spoiler surface also had sharp edges and corners, which made it more susceptible to flow separation and turbulence, which increased drag. As shown in Fig 8. Even though the side mirrors and roof spoiler individually contributed a modest amount of drag, their combined effect produced significant drag forces and the creation of high-pressure zones.

The frontal area is occupied by static pressure, which creates a significant barrier to the vehicle's forward motion. Because of stagnation, the bus's frontal area is under the most pressure. On the side of the vehicle, or the surface parallel to the flow, a vacuum is produced because the static pressure occupies much of the airspace. Including a smaller pressure range in a vehicle’s design can help lower drag force and increase fuel economy (Akmal Bin Nasrul Hisham, 2008; Nikam & Borse, 2014). According to Table 2, the 1st model’s pressure range is 8870 Pa. In contrast, the 2nd model’s pressure range is 6594 Pa and 7070 Pa (with windows) whereas the 3rd model’s pressure range is 6751 Pa. Therefore, it is evident that the 2nd model has the lowest pressure-range value which reduces the drag force and improves the fuel efficiency.

5.2. Velocity Contours

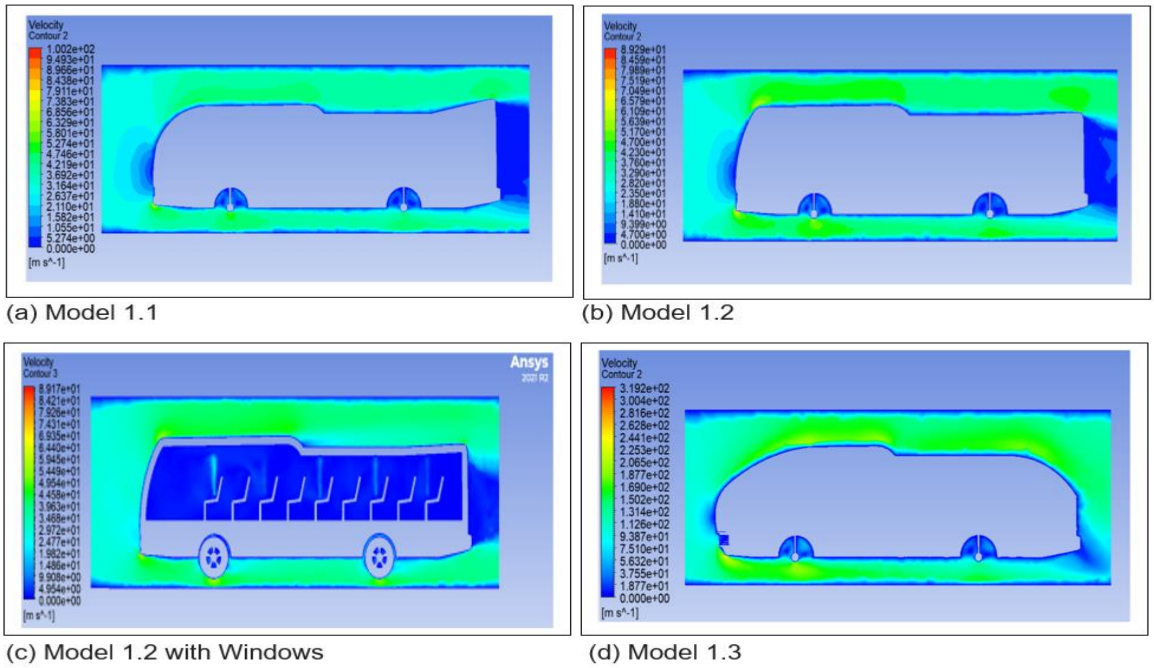
As the flow crosses the edges of the front and rear faces, its velocity increases correspondingly. This can be attributed to the front face. The flow is not exposed directly on the road wheels because side wings are present. Because of the filleted edge at the back, the flow direction is diverted there. As a result, the vehicle's rearward vacuum zone is under control.

Table 2 Pressure values obtained after the analysis.

Boundary	Highest Pressure	Lowest pressure
Model 1.1	1762 Pa	-7108 Pa
Model 1.2	1110 Pa	-5484 Pa
Model 1.3	1403 Pa	-5348 Pa
Model 1.2 with windows	1158 Pa	-5912 Pa

Table 3 Accumulated values from the velocity contour

Model	Highest velocity	Lowest velocity
Model 1.1	100.2 m/s	5.274 m/s
Model 1.2	89.29 m/s	4.7 m/s
Model 1.3	319.2 m/s	18.7 m/s
Model 1.2 with windows	89.17 m/s	4.954 m/s



(a) Model 1.1 (b) Model 1.2

(c) Model 1.2 with Windows (d) Model 1.3

Fig. 9 Velocity contour of the models

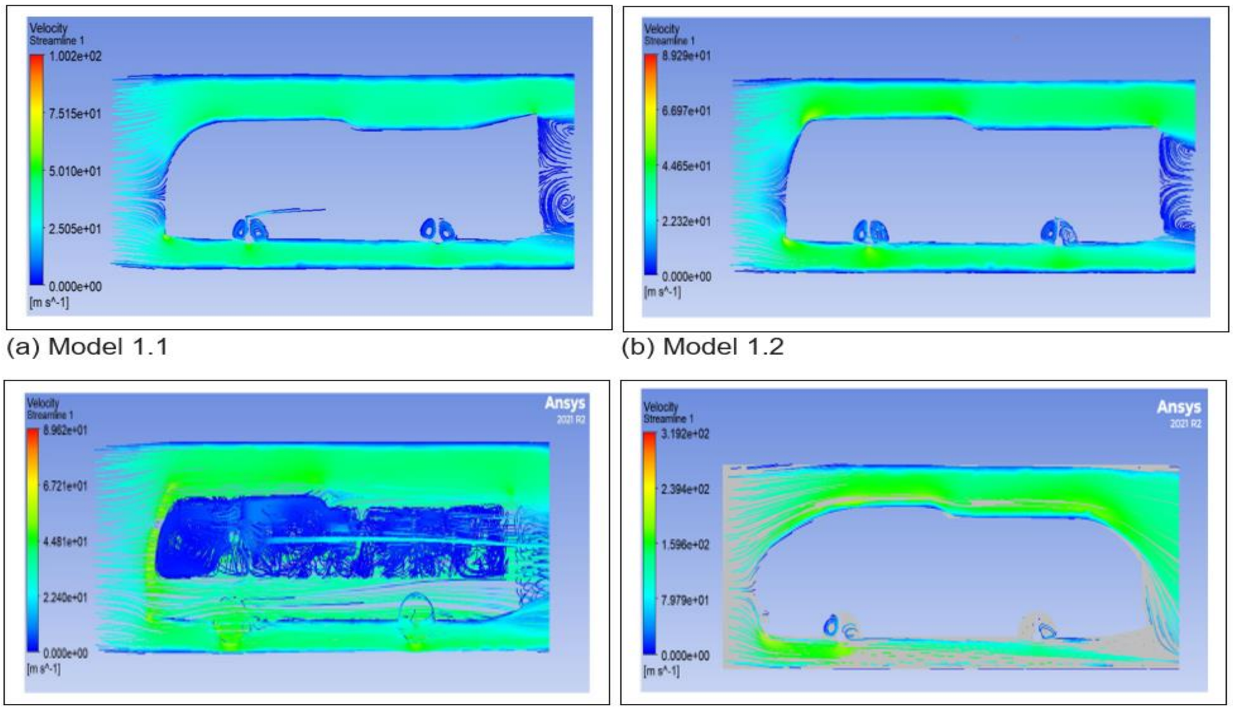
Evidently, from Table 3 the 1st model consists of a velocity range of about 94.926 m/s, and for the 3rd model 300.2 m/s. The 2nd model range is 84.59 m/s and 84.216 m/s (with windows). Seemingly, it is conceivable that the 1st and 3rd model comprises a greater velocity range than that of the 2nd model. The high-velocity range causes flow separation, recirculation, and high-speed flow which increases drag force, reduces fuel efficiency, and generates noise. So, it is deemed that the 2nd model is the best aerodynamic model.

5.3. Velocity streamline

All of the velocity contours suggest that as the air gets closer to the bus's front, its velocity drops. then the airspeed moves further away from the front of the bus. According to the vector plot of (Nikam & Borse, 2014) both the magnitude of K.E. and the intensity of turbulence at the top of the bus increase with increasing speed. Similarly, in our analysis, the velocity streamlines and contour plot suggest that air is attempting to enter through the front windows and move downstream. At the front radius area, the flow accelerates and becomes stagnant at the frontal area. There was evidence of flow separation behind the mirrors. The roof and sides of the flow continue to be connected. Eventually, the flow separates at the back and creates vortices. The rotating wheel region was another significant flow separation area where vortices generation was noted.

Velocity streamlines generally depict the direction and magnitude of velocity. Analyzing the velocity streamlines, the 1st and 2nd models have similar streamlines. The flow at the end of the bus is lower than the front. However, the 2nd model has a lower velocity range than the 1st model. On the other hand, the 2nd model with windows has dispersed velocity streamlines due to windows. The velocity range is almost like the 2nd model. Lastly, the 3rd model has streamlines that are different from others due to the shape of the end of the bus. The velocity streamline fully depends upon the shape of the bus.

The streamlines in Fig. 10 show how the air passes over the automobiles. Compared to the other bus models,



(a) Model 1.1 (b) Model 1.2

(c) Model 1.2 with Windows (d) Model 1.3

Fig. 10 Velocity streamlines of the models

the airflow of Model 1.2 is smoother in the case of closed windows. The aerodynamic performance of the vehicle can be effectively enhanced by a streamlined front end that facilitates better airflow, highlighting potential areas for improvement in the new design to achieve better aerodynamics.

5.4. Theoretical Fuel Consumption

Marais et al. (2010) conducted a study to offer an analysis for assessing how well electrical air conditioning systems operate in cars with conventional combustion engines. He discovered that using the air conditioning increased overall consumption by 0.4 L/100 km. When the outcomes of the five simulated environmental scenarios are combined, the average increase in total fuel consumption over the NEDC period is 0.96 L/100 km. Pesaran et al. (1992) presented in his study that the average fuel consumption of a bus engine is approximately 0.315 L/kWh. Weight is another factor in fuel consumption. Fuel consumption rises by l% for every 2% increase in vehicle weight. Zhang et al. (2014) provides a comprehensive study where the addition of an air compression system increases fuel consumption. However, at average speeds of up to 25–30 km/h, the increased percentage was only 3%. This is because as average speed increased, average fuel consumption rates would progressively rise, increasing the percentage of the drivetrain system's energy demand. According to (Zulkifli et al., 2015) An air conditioning system can affect a car's fuel consumption by as much as 20%. In their experiment, there is a 5–14% difference in fuel reduction between the compressors that are driven by electricity and belts. Abinesh and Arunkumar (2014) altered the bus's exterior and structural features aerodynamically to lessen the vehicle's drag force, which lowers the vehicle's fuel consumption. There is a 10% overall decrease in aerodynamic drag force. When the speed was 100 kmph, the modifications resulted in a 16.33% fuel reduction. Muthuvel et al. (2013) modified the bus's exterior and structural features aerodynamically to lessen the vehicle's drag force, which lowers the fuel consumption of the vehicle. Implementing the new design, there has been a significant drop in drag force of roughly 30%–34%, and every 100 kilometers, 6–7 liters of fuel are saved. Damissie and Ramesh Babu (n.d.) conducted a study to reduce the drag force on the bus, which lowers the bus's fuel consumption, by altering the aerodynamic outer surface of the vehicle. The test results clearly showed that the new concepts have a significant reduction in drag force compared to the existing bus, of approximately 39.4%, and that every 100 kilometers, 22.8% (5 to 6 liters) of fuel was saved

During this particular analysis to make the calculation more efficient, all the bus models are assumed to have the same engines and other components. Therefore, the overall efficiency of the drivetrain is taken at 0.37. The heating value and the density of the fuel are 44000 kJ/kg and 0.82 kg/L. Considering the bus that travels 500 km per day.

A moving vehicle's drag force is calculated using

$$F = \frac{1}{2}\rho A V^2 Cd \tag{10}$$

Work done due to drag force, $W = FL$ (11)

Energy input, $E = \frac{W}{n}$ (12)

Amount of fuel $M = \frac{E}{HP}$ (13)

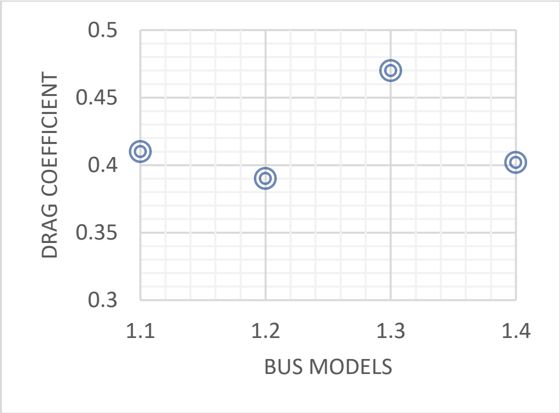


Fig. 11 Result obtained after the CFD analysis

Here, F, L, n, H, and P represent Drag force, Distance traveled, overall efficiency, heating value, and fuel density respectively.

Now, materializing equations (10), (11), (12) and (13)

- ✓ Fuel consumption by Model 1.1 is 196.15 Liters.
- ✓ Fuel consumption by Model 1.2 is 179.98.
- ✓ Fuel consumption by Model 1.3 is 216.86 Liters.
- ✓ Fuel consumption by Model 1.2 with windows 184.43 Liters.

If Model 1.2 is converted into non-ac by adding windows, the fuel consumption gets increased to 184.43 Liters which is 2.5% higher due to the extra drag induced by the windows. By mobilizing the shape from Model 1.2, a bus can save around 16.2 liters of fuel per 500 km. The engine directly impacts the operation of an AC system. Because the engine drives the AC compressor, there is a strong correlation between fuel efficiency, air conditioning, and AC run time. When the AC compressor is turned on, more fuel is used in this situation. Due to this, every 100 kilometers, 0.2-1 liters of fuel are used (Pesaran et al., 1992; Marais et al., 2010; Zhang et al., 2014; Zulkifli et al., 2015). This acknowledges that an AC system comprising closed-window buses will also be cost-effective for intercity passenger transportation.

As per the demonstrations of Fig. 11 Model 1.2 has approximately 5% less drag coefficient than Model 1.1. Model 1.2 with windows (shown as 1.4 in Fig. 11) has a 3% higher drag coefficient than Model 1.2 without windows. During the analysis, the fuel consumption per 500 Km for all models was also evaluated. The buses with side windows consume slightly more fuel than buses without side windows. Model 1.2 reduces fuel consumption by 8.3% compared to Model 1.1. It gets to 17% compared to Model 1.3. Model 1.2 with side windows consumes around 2.5% more fuel than the same model without side windows.

5.5. Validation of the Bus Model

This investigation is validated based on the study of Damissie and Ramesh Babu (n.d.). Utilizing the RMSE (Root Mean Square Error) method 10.41% deviations of C_d were found when the bus models were operated at 100 km/h. Another validation is attained compared to (Bayindirli, 2019) This study concentrated on the passive flow canal. where the simulation results were in good agreement with the wind tunnel results. The current research consists of 10.5% deviations compared to this particular study.

$$RMSE=\sqrt{(\frac{\sum Error^2}{n})} \tag{14}$$

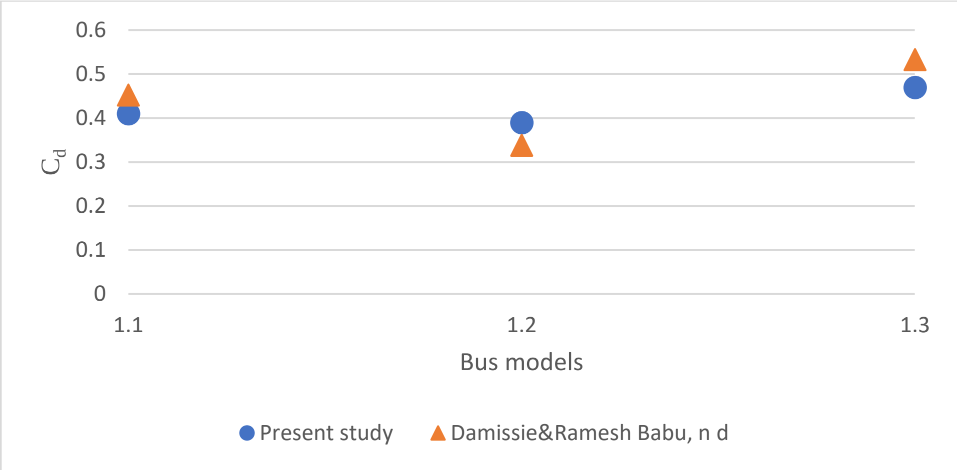
5.6. Aerodynamic Evaluation of the New Design

The regions where the airflow meets the least resistance or where there are notable geometric features that cause turbulence or acceleration of the air are usually the ones with the highest air velocity in a Computational Fluid Dynamics (CFD) airflow simulation over a bus.

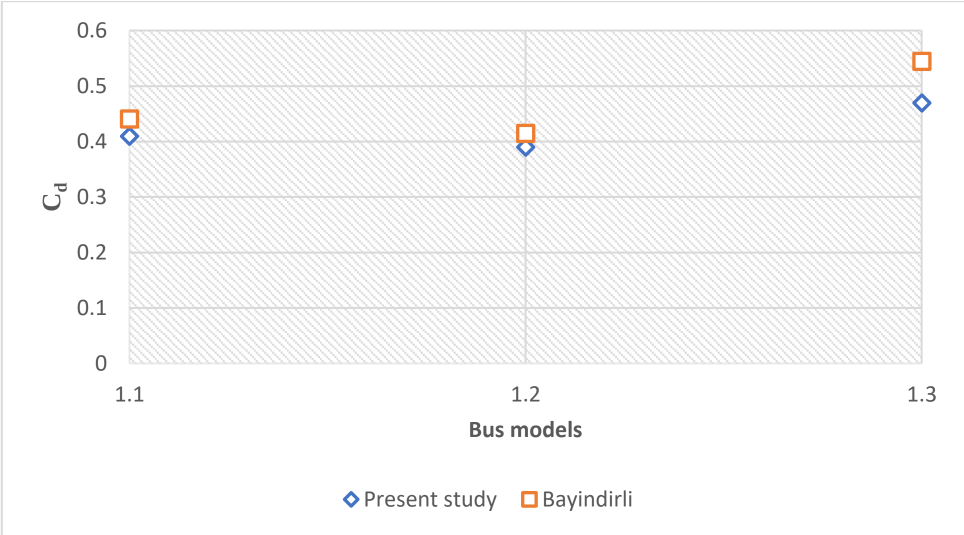
Since boundary layers separate and vortices form around sharp edges and corners, air tends to accelerate there. It is evident in Fig. 9 & 10. In front of the front wheel underneath the bus, there's a sharp edge where the air gets accelerated and maximum velocity is achieved. Therefore, in the bus models, the edge in front of the front wheel causes airflow acceleration and consequently, the highest velocity, and the rear end of the bus edge feels the lowest air velocity. Evidently from all the bus model's pressure contours, the front face of the bus models experiences the maximum pressure and the rear face experiences the least pressure.

The current analysis reveals the drag forces, drag coefficients, and the percentage variation of the fuel consumption for different bus models. Model 1.2 serves as the baseline as it has the lowest drag force of 4805 N and a drag coefficient of 0.39. Comparing the other models to Model 1.2, interesting insights can be derived. The drag force and drag coefficient of Model 1.1 were found to be 5237 N and 0.41 which results in a 9% increase in fuel consumption compared to Model 1.2. Model 1.3 shows a significantly higher drag force and drag coefficient of 5789.9 N and 0.47. This results in a notable 20.50% increase in fuel consumption compared to Model 1.2. Finally, Model 1.2 with windows (shown as 1.4 in Fig. 13) has a drag force of 4924 N and a drag coefficient of 0.402, leading to a 2.5% increase in fuel consumption compared to the windowless Model 1.2.

In this project, we have analyzed 3 different designed intercity bus models for reducing aerodynamic drag and fuel consumption. The results of the CFD analysis are investigated including the coefficient of drag, pressure, and velocity for these bus models. These findings indicate that our modified vehicle that is Model 1.2 produces less aerodynamic drag than the designs that are currently in use. A vehicle's aerodynamic shape is important since it significantly impacts fuel efficiency. It can be assured that intercity transportation bus models with closed windows that comprise AC will be more efficient in fuel consumption. Thus, it is important to understand the effects of aerodynamic features when purchasing a new vehicle to ensure that the time and money spent in this area will be worthwhile.



(a) Validation against (Damissie & Ramesh Babu, n.d.)



(b) Validation against (Bayindirli, 2019)

Fig. 12 Drag coefficient of the bus models compared to experimental studies

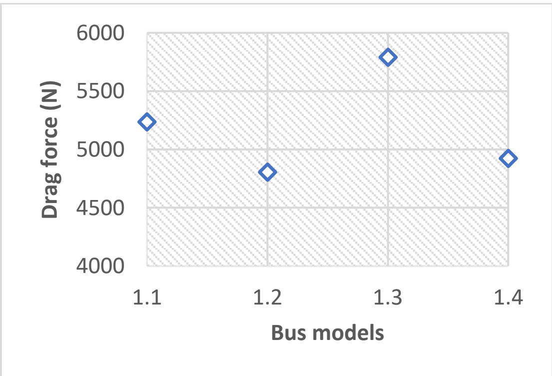


Fig. 13 Drag force value accumulated from the theoretical calculation

6. CONCLUSION

This study provides in-depth knowledge of aerodynamic analysis and passenger bus shape modifications. During the study, interesting trends were observed in fuel consumption per 500 km for different bus models. This indicates that the shape modifications made to the bus have reduced fuel consumption. The findings validate the research's objective of improving fuel efficiency through aerodynamic analysis and shape modifications. The significant decrease in fuel consumption observed in Model 1.2 emphasizes the positive impact of optimized aerodynamics. Model 1.2 without windows demonstrates a fuel consumption of 180 liters per 500 km. At the same time, Model 1.2 with windows exhibits a slightly higher fuel consumption indicating a 2.5% increase. These findings suggest that windows in non-ac buses negatively impact fuel efficiency. Model 1.2 represents the most significant improvement observed in the current research, with a notable 7.7% reduction in drag force when compared to this base model. Additionally, Model 1.2 with open windows shows a commendable 5.4% reduction in drag force over the base model. Considering the perspective of Bangladesh's road transportation, the investigation provides valuable insights into the effects of aerodynamic optimization on fuel efficiency and drag coefficient. The study also contributes to the field of transportation by highlighting the importance of aerodynamic optimization. Better aerodynamic shape can enhance fuel efficiency and reduce operational costs. The paper serves as a valuable resource for further research and development in the design and modification of buses to achieve better aerodynamic performance and improved sustainability. However further study needs to be done extensively on many other aspects. This includes passenger comfort, structural integrity, or cost implications of implementing the shape modifications. These factors are also crucial considerations in real-world scenarios and play significant roles in practical applications.

ACKNOWLEDGEMENTS

We would like to thank the Department of Mechanical Engineering, Chittagong University of Engineering and Technology for their unwavering support and valuable resources that made this research possible. Their commitment to cultivating an environment of academic excellence and innovation has played an important role in shaping the outcome of this study.

Additionally, we would like to express our gratitude to Aftab Motors Company Limited for their cooperation and kind assistance during this research. Being able to lend us access to their facilities and technical knowledge greatly aided in the data collection process and improved our comprehension of the topic.

CONFLICT OF INTEREST

The authors declare that there is no potential conflict of interest.

AUTHORS CONTRIBUTION

Mrinmoy Roy Rony: Conceptualization, Data curation, Visualization, Writing - Original Draft, Formal analysis, Methodology, Validation. **Md Jobayer Islam:** Conceptualization, Data curation, Formal analysis, Investigation, Methodology, Software. **Shihab Shahriare:** Prototype design, Software, Data curation. **Md. Mahbubul Alam:** Project administration Supervision, Validation, Writing – review & and editing

REFERENCES

Abinesh, J., & Arunkumar, J. (2014). CFD analysis of aerodynamic drag reduction and improve fuel economy. *International Journal of Mechanical Engineering and Robotics Research, 3*(4), 430-440. www.ijmerr.com

Akmal Bin Nasrul Hisham, A. (2008). *Study and Improvement of Drag Coefficient of a Bus*. [BSc Thesis, Universiti Teknologi Petronas], Perak.

Alexei Pichardo-Orta, F., Luna, O. A. P., & Cordero, J. R. V. (2022). A frontal air intake may improve the natural ventilation in urban buses. *Scientific Reports*, *12*(1). https://doi.org/10.1038/s41598-022-25868-x

Alonso-Estébanez, A., Del Coz Díaz, J. J., Álvarez Rabanal, F. P., & Pascual-Muñoz, P. (2017). Numerical simulation of bus aerodynamics on several classes of bridge decks. *Engineering Applications of Computational Fluid Mechanics*, *11*(1), 435–449. https://doi.org/10.1080/19942060.2016.1201544

Al-Saadi, A., Al-Farhany, K., Idan Al-Chlaihawi, K. K., Jamshed, W., Eid, M. R., Tag El Din, E. S. M., & Raizah, Z. (2022). Improvement of the aerodynamic behavior of a sport utility vehicle numerically by using some modifications and aerodynamic devices. *Scientific Reports*, *12*(1). https://doi.org/10.1038/s41598-022-24328-w

Amen Mohamed, E., Naeem Radhwi, M., & Farouk AbdelGawad, A. (2015). Eyad amen mohamed, muhammad naeem radhwi, ahmed farouk abdelgawad. computational investigation of aerodynamic characteristics and drag reduction of a bus model. *American Journal of Aerospace Engineering. Special Issue: Hands-on Learning Technique for Multidisciplinary Engineering Education*, *2*(1), 64–73. https://doi.org/10.11648/j.ajae.s.20150201.16

Arteaga, O., Hernán, M. V., Terán, H., Chacon, S., Lara, M. A., Rocha-Hoyos, J., Arguello, E., & Aguirre, R. P. (2020). *Aerodynamic optimization of the body of a bus.* IOP Conference Series: Materials Science and Engineering. https://doi.org/10.1088/1757-899X/872/1/012002

Bayindirli, C. (2019). Drag reduction of a bus model by passive flow canal. *International Journal of Energy Applications and Technologies*, *6*(1), 24–30. https://doi.org/10.31593/ijeat.533745

Belachew, B., Nallamothu, R. B., Nallamothu, S. K., & Nallamothu, A. K. (2021). Improving frontal body shape of locally built fsr isuzu bus for reduction of aerodynamic resistance. In K. Jha, P. Gulati & U. K. Tripathi (Eds.), *Recent Advances in Sustainable Technologies* (pp. 323–338). Lecture Notes in Mechanical Engineering. Springer, Singapore. https://doi.org/10.1007/978-981-16-0976-3_31

Damissie, H. Y., & Ramesh Babu, N. (n.d.). Aerodynamic drag reduction on locally built bus body using computational fluid dynamics (CFD): A case study at bishoftu automotive industry. *International Journal of Engineering Research & Technology, 6*(11), 276-283. www.ijert.org'

Energy Agency, I. (2021). *Review 2021 Assessing the effects of economic recoveries on global energy demand and CO 2 emissions in 2021.* Global Energy. www.iea.org/t&c/

Garcia-Ribeiro, D., Bravo-Mosquera, P. D., Ayala-Zuluaga, J. A., Martinez-Castañeda, D. F., Valbuena-Aguilera, J. S., Cerón-Muñoz, H. D., & Vaca-Rios, J. J. (2023). Drag reduction of a commercial bus with add-on aerodynamic devices. *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile*

Engineering, *237*(7), 1623–1636. https://doi.org/10.1177/09544070221098209

Gohari, M., Norozi, R., & Aghdam, A. H. (2022). Evaluation and optimization of the aerodynamic noise reduction of vehicle side view mirrors: experimental and numerical study. *HighTech and Innovation Journal*, *3*(1), 73–84. https://doi.org/10.28991/HIJ-2022-03-01-08

Government of the People's Republic of Bangladesh, Ministry of Communications, Roads and Railways Division (1999) *Roads in Bangladesh The Next Millennium*. https://www.rhd.govt.bd

Government of The People's Republic of Bangladesh, Ministry of Road Transport and Bridges, Road Transport and Highways Division. (2016). Regional Road Connectivity Bangladesh Perspective. https://www.rhd.govt.bd

Hucho, W. H., & Sovran, G. (1993). Aerodynamics of road vehicles. *Annual Review of Fluid Mechanics*, *25*, *485-537*. *https://doi.org/10.1146/annurev.fl.25.010193.002413*

Ince, I. T. (2010). *Aerodynamic analysis of GTD model administrative service vehicle* [PhD Thesis, Gazi University Institute of Science], Ankara.

Jadhav, C. R., & Chorage, R. P. (2020). Modification in commercial bus model to overcome aerodynamic drag effect by using CFD analysis. *Results in Engineering*, *6*. https://doi.org/10.1016/j.rineng.2019.100091

Marais, C., Holdack-Janssena, H., & van Niekerkb, T. (2010). Fuel consumption for conventional and electrically driven ac compressors. *R & D Journal of the South African Institution of Mechanical Engineering*, *26*, 21-26. http://www.saimeche.org.za

Mezarciöz, S. (2010). *Aerodynamics of a model bus* [MSc Thesis, Çukurova University Institute of Natural and Applied Sciences], Adana

Mukut, A. N. M. M. I., & Abedin, M. Z. (2019). Review on aerodynamic drag reduction of vehicles. *International Journal of Engineering Materials and Manufacture*, *4*(1), 1–14. https://doi.org/10.26776/ijemm.04.01.2019.01

Muthuvel, A., Murthi, M. K., Sachin N P, Koshy, V. M., Sakthi, S., & Selvakumar, E. (2013). Aerodynamic exterior body design of bus. *International Journal of Scientific & Engineering Research*, *4*(7). http://www.ijser.org

Nikam, K. C., & Borse, S. L. (2014). Study of Air Flow Through a Open Windows Bus Using OpenFOAM. *International Journal of Fluids Engineering* 6(1), 54-64. http://www.irphouse.com

Oforji, P. I., Mba, E. J., & Okeke, F. O. (2023). The effects of rhythm on building openings and fenestrations on airflow pattern in tropical low-rise residential buildings. *Civil Engineering Journal (Iran)*, *9*(8), 2062–2084. https://doi.org/10.28991/CEJ-2023-09-08-016

Pesaran, A. A., Parent, Y. O., & Bharathan, D. (1992). Non-CFC air conditioning for transit buses. *SAE Technical Paper 922459.* https://doi.org/10.4271/922459

Pfeiffer, I. F., & Wriggers, I. P. (n.d.). *lecture notes in applied and computational mechanics.* Series Editors.

Rashwan, A. (2019). Topology optimization and rim design. *International Journal of Engineering and Management Sciences (IJEMS)*, *4*(4). https://doi.org/10.21791/IJEMS.2019.4.10

Raveendran, A., Sridhara, S. N., Rakesh, D., & Shankapal, S. R. (2009). *Exterior styling of an intercity transport bus for improved aerodynamic performance*. SAE Technical Paper 2009-28-0060. https://doi.org/10.4271/2009-28-0060

Shekar, V. T., Thuraimoni, A., Reddy, S., & Cherukuri, A. (2014). A holistic approach to aerodynamics of intercity and interurban buses. *SAE Technical Papers*, *1*. https://doi.org/10.4271/2014-01-0581

Tefera, A., Bekele, A., & Pandey, V. (2023). Investigating the aerodynamic effect of rooftop cargo compartment - a case of modified intercity bus. *Engineering Research Express*, *5*(3). https://doi.org/10.1088/2631-8695/acf54a

Thomas, M., Sharma, R. N., & Kilduff, M. (2006). Aerodynamic Effects of Different Ventilation Methods on Buses. In *Fourth International Symposium on Computational Wind Engineering (CWE)*, Yokohama. https://iawe.org'

Thorat, S., Amba, G., & Rao, P. (2011). Computational analysis of intercity bus with improved aesthetics and aerodynamic performance on Indian roads. *International Journal of Advanced Engineering Technology*, *2*(3),103-109. E-ISSN 0976-3945.

Ulum, M. S., Arminda, W., Kamaruddin, M., & Satria, W. D. (2023). Ventilation performance of air duct in double loaded corridor building: a case study. *Civil Engineering Journal.* *9*(10), 2445-2455. https://civilica.com/doc/1962978

Vinayagam, P., Rajadurai, M., Balakrishnan, K., & Priya, G. M. (2017). Design modification on indian road vehicles to reduce aerodynamic drag. *International Journal of Advanced Engineering, Management and Science*, *3*(8), 850–854. https://doi.org/10.24001/ijaems.3.8.6

Wisconsin K-12 Energy Education Program (2020). *Facts about Oil. Energy Resource Fact Sheets. http://keepprogram.org*

Wood, R., & Bauer, S. (2003). Simple and low-cost aerodynamic drag reduction devices for tractor-trailer trucks. *SAE Technical Paper* 2003-01-3377, https://doi.org/10.4271/2003-01-3377

Yelmule, M. M., Kale, S. R., & Veeravalli, S. V. (2009). Aerodynamics of a bus with open windows. *International Journal of Heavy Vehicle Systems*, *16*(4), 459-488. https://doi.org/10.1504/IJHVS.2009.027415

Yudianto, A., Adiyasa, I. W., & Yudantoko, A. (2021). Aerodynamics of bus platooning under crosswind. *Automotive Experiences*, *4*(3), 119–130. https://doi.org/10.31603/ae.5298

Zhang, S., Wu, Y., Liu, H., Huang, R., Yang, L., Li, Z., Fu, L., & Hao, J. (2014). Real-world fuel consumption and CO2 emissions of urban public buses in Beijing. *Applied Energy*, *113*, 1645–1655. https://doi.org/10.1016/j.apenergy.2013.09.017

Zulkifli, A. A., Dahlan, A. A., Zulkifli, A. H., Nasution, H., Aziz, A. A., Perang, M. R. M., Jamil, H. M., & Misseri, M. N. (2015). *Impact of the electric compressor for automotive air conditioning system on fuel consumption and performance analysis.* IOP Conference Series: Materials Science and Engineering, https://doi.org/10.1088/1757-899X/100/1/012028