Advanced and Sustainable Technologies
ASET
E-ISSN 2976-2294

Volume 3, No 1, June 2024 [80-88]

Airflow Analysis of the Bus Under Various Velocities Using CFD Simulation

Muhammad Aiman Hakim Samsuddin[1*]

[1]Faculty of Mechanical Engineering & Technology, Universiti Malaysia Perlis (UniMAP), 02600 Arau, Perlis, Malaysia

Received 21 March 2024, Revised 19 April 2024, Accepted 22 April 2024

ABSTRACT

This project investigated the aerodynamic effects of various airflow velocities around a bus. Key parameters included airflow velocity and turbulence models, with dependent variables being the velocity and pressure on the bus body. The bus model, sourced from an existing GrabCAD file, was imported into SimFlow 4.0 for analysis. A steady-state simulation was employed, incorporating symmetry conditions, turbulence modeling, boundary conditions, and a baseline post-processing method to visualize the velocity and pressure distributions around the bus. Mesh refinement was carefully adjusted to accommodate the bus's large size and accurately capture the flow gradients. The simulation results were analyzed using Paraview post-processing to evaluate the final velocity and pressure contours. The results showed that the airflow velocities crucially affect the velocity and pressure distribution around the bus. This study yielded significant and insightful results by varying the independent parameters, advancing the understanding of bus aerodynamics under different airflow velocities.

Keywords: Bus, SimFlow 4.0, Turbulent model, Steady-state simulation, Computational Fluid Dynamics.

1. INTRODUCTION

Aerodynamics, a branch of physics, examines the motion of fluids and their interaction with objects moving through them [1]. This project focuses on the aerodynamic behavior of a bus body subjected to various airflow velocities. Understanding aerodynamics is crucial for controlling the maneuverability of the bus, particularly in challenging conditions such as storms. Additionally, this knowledge is vital for designing and constructing buses using materials capable of withstanding the pressures exerted by high-speed winds. Several factors influence the aerodynamic properties of vehicles, including shape, size, surface texture, viscosity, and fluid density [2]. These factors significantly impact how different vehicles, such as cars and buses, interact with airflow. For instance, sports cars are typically small and feature a sleek, pointed front design that minimizes air resistance, allowing them to travel at high speeds easily [3]. In contrast, buses have large bodies and expansive frontal areas, which result in higher air resistance and limit their ability to cruise at high speeds.

The design differences between cars and buses are primarily driven by their intended functions. Sports cars are engineered for speed and agility, optimized to travel from point A to point B in the shortest possible time, albeit with limited passenger capacity. Conversely, buses are designed to transport large numbers of passengers efficiently [4]. However, their large size and shape create significant aerodynamic drag, hindering their speed capabilities [5]. Analyzing buses' aerodynamic performance under various airflow conditions using Computational Fluid Dynamics

*Corresponding author: aimankhair1225@gmail.com

(CFD) simulations provides deeper insights into the factors affecting bus aerodynamics [6]. This understanding can lead to design improvements that enhance bus efficiency, safety, and performance, ultimately contributing to better public transportation systems. In automotive aerodynamics, understanding airflow behaviour around vehicles is crucial for optimizing performance, enhancing fuel efficiency, and ensuring safety [7]. Due to their large size and unique geometry, buses present distinct aerodynamic challenges [8] that differ significantly from those encountered in smaller vehicles.

Several studies [9, 10, 11, 12] have been conducted using the Computational Fluid Dynamics (CFD) method to analyze, improve, and optimize the aerodynamic aspects of buses. These studies have contributed to understanding how aerodynamic design affects fuel efficiency, stability, and overall performance. Jadhav and Chorage [9] utilized CFD to evaluate the impact of modification of commercial buses on the drag coefficient for a typical coach bus. The drag on the existing bus body shows only slight variations with speed changes. In contrast, the modified bus body reduces drag as speed increases. At higher speeds, the air flows more smoothly over the modified bus body, with improved guidance and reduced turbulence compared to lower speeds. In the automotive industry, drag reduction systems have significantly decreased fuel consumption, meeting environmental regulations and consumer demands. While extensive research has focused on reducing aerodynamic drag for light and heavy vehicles, buses have been largely overlooked.

Daniel et al. [10] examined the airflow over the Marcopolo Paradiso 1200 G7 commercial bus and the impact of three aerodynamic devices—vortex generators, lateral devices, and rails—on drag force and fuel consumption. Using wind tunnel tests and Computational Fluid Dynamics simulations, 11 different device arrangements were evaluated. Results showed a maximum reduction of 8.63% in the drag coefficient and 3.92% in fuel consumption. These initial findings suggest that further optimization could lead to even greater efficiencies, with potential applications across other commercial buses. Besides, reducing aerodynamic drag could save fuel consumption [11]. In the study by Kanekar et al. [12], CFD-driven alterations to the bus design yield a notable 28% enhancement in the drag coefficient. This results in a corresponding 20% increase in fuel efficiency, facilitated by the improvements driven by CFD analysis [12].

Thus, this study aims to investigate the effects of varying airflow velocities on the aerodynamic characteristics of a bus. The research aims to uncover critical insights into the flow dynamics by analysing key parameters such as velocity and pressure distributions. CFD simulations offer a powerful tool for visualizing and quantifying these effects, enabling detailed examination without costly and time-consuming physical experiments. The bus model, imported from an existing design, serves as the basis for the simulations, ensuring realistic and applicable results.

2. MATERIAL AND METHODS

In this study, SimFlow 4.0 software [13] was utilized for mesh generation and simulation to analyze airflow dynamics around the bus. The simulation focuses on conducting an external aerodynamic analysis, emphasizing a steady-state evaluation of the bus body's aerodynamic performance. Figure 1 depicts the geometric representation of the bus utilized in the current investigation. Symmetry conditions were incorporated to streamline computations and reduce the computational domain's size. Turbulence modeling, boundary conditions, and baseline post-processing features were meticulously defined within SimFlow. Various inlet velocities, specifically 15 m/s, 30 m/s, and 40 m/s, were considered and implemented. In the meshing process, refinement levels near the bus geometry were adjusted, with minimum and maximum refinement levels set at 2 and 3, respectively, to ensure the total element count remained below 200k nodes. Mesh divisions of (40, 12, 15) were applied along the x-, y-, and z-axes of the flow domain, with corresponding cell sizes of 0.45 m, 0.33 m, and 0.27 m, as illustrated in Figure 2.

Figure 3 shows the hexahedral dominant meshing element of the domain and bus model. Boundary conditions were defined across domain surfaces, including inlet velocity (X-), pressure outlet (X+), symmetry plane (Z-), and wall boundary (Y-). All simulations were conducted under steady-state conditions, assuming incompressible flow. A SIMPLE discretization solver was implemented within SimFlow to ensure numerical stability and accuracy. In the first analysis, the inlet velocity varies from 15 m/s, 30 m/s, and 40 m/s, using the k-ω SST model. The second analysis uses the different turbulent models RANS (k-ε) and RANS (k-ω SST) at an inlet velocity 5 m/s.

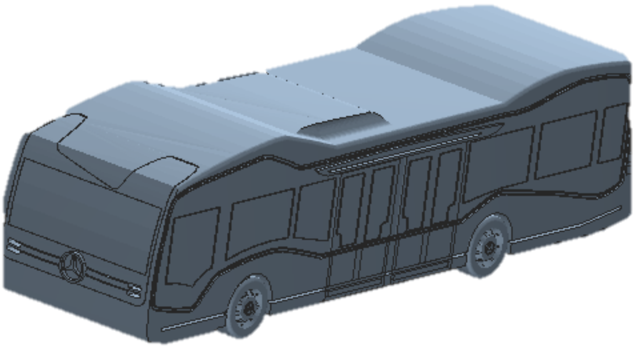


Figure 1: Geometry of bus body.

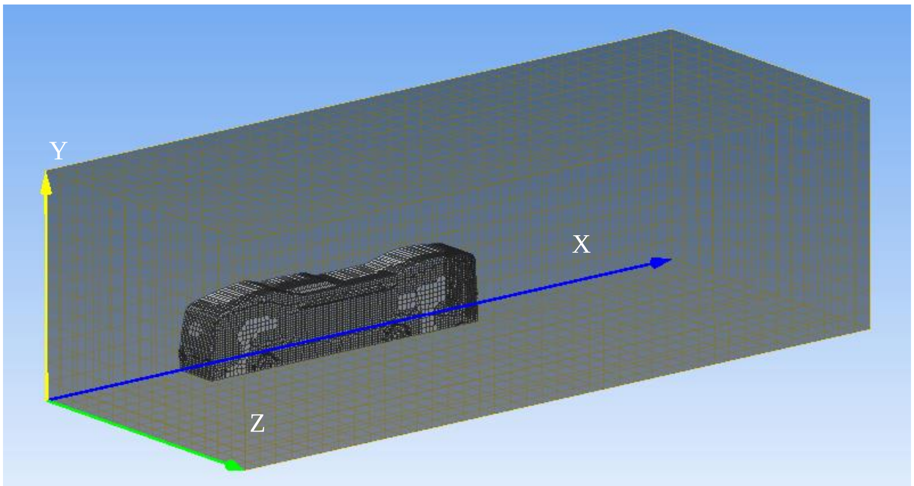
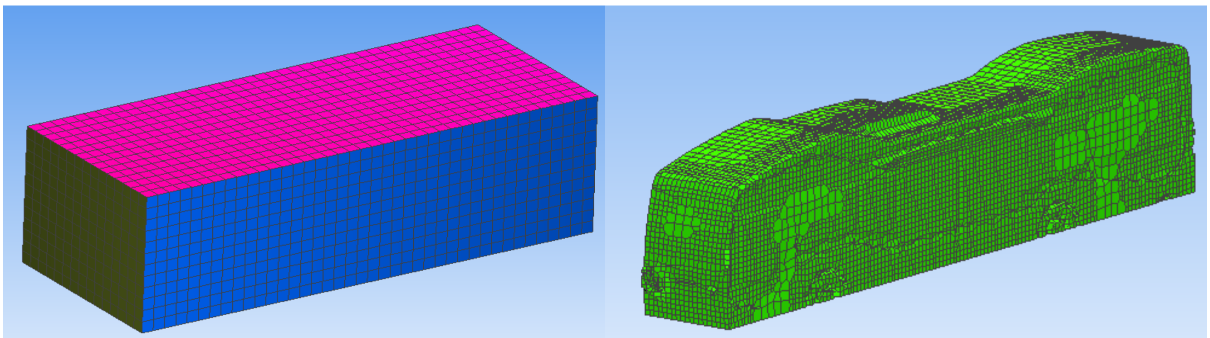


Figure 2: Symmetrical meshed model of the bus.



(a) (b)

Figure 3: (a) meshing of the domain and (b) meshing on the bus body.

3. RESULTS AND DISCUSSION

Throughout the execution of this experiment, numerous challenges and obstacles arose, necessitating solutions at every stage, from importing the STL file to visualizing the results in Paraview. Initially, aligning the bus body posed a significant challenge due to the substantial dimension difference. Analyzing the bus body proved complicated, requiring adjustments to simulation settings to ensure accuracy. The bus body was sourced from the GrabCAD website, where all CAD files are available, necessitating scaling to maintain realistic proportions relative to the car.

The results revealed a clear trend of direct proportionality when comparing each inlet velocity with the maximum velocity. As the inlet velocity increased, so did the maximum velocity, indicating a linear relationship between the two variables (Figure 4). Similarly, the graph depicted a direct correlation between inlet velocity and maximum pressure, with higher inlet velocities corresponding to elevated maximum pressures (Figure 5). This finding suggests that higher inlet velocity exerted more pressure on the bus, potentially impacting its structural integrity. These observations highlight the significant influence of airflow dynamics, particularly during intense weather conditions [14], on the bus's aerodynamic performance and structural resilience.

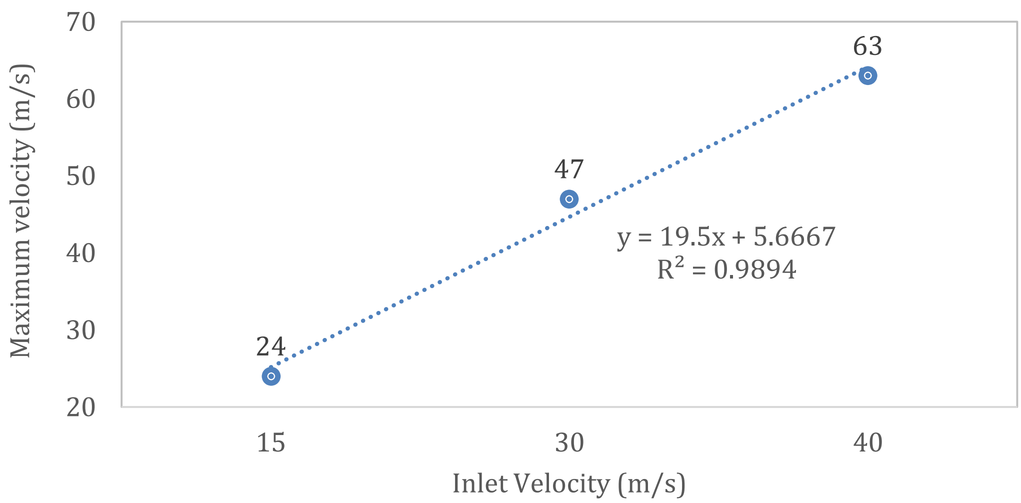


Figure 4: Inlet velocity vs maximum airflow velocity over the bus.

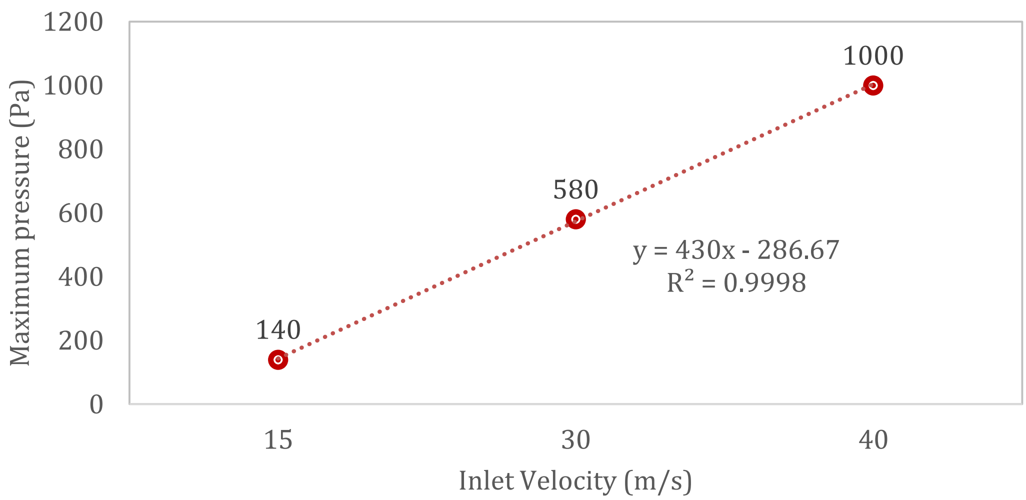


Figure 5: Inlet velocity vs maximum airflow pressure over the bus.

The simulation analysis revealed that at an inlet velocity of 15 m/s, the maximum velocity recorded was 24 m/s. Notably, the red region at the top of the bus indicated areas of high velocity. In contrast, the lowest velocity, nearly 0 m/s, was registered at the lower front and rear of the bus (Figure 6). Regarding pressure distribution, the highest pressure recorded was 14 Pa, with the red region at the front of the bus indicating high-pressure areas (Figure 7). However, most areas around the bus exhibited a pressure of 0 Pa, as denoted by the yellow region. However, certain specific points, such as the tip of the upper front, showed a blue region indicating negative pressure, valued at -320 Pa. These findings provide detailed insights into the airflow dynamics and pressure distribution around the bus under specific inlet velocity conditions. A similar phenomenon is observed at other inlet velocities.

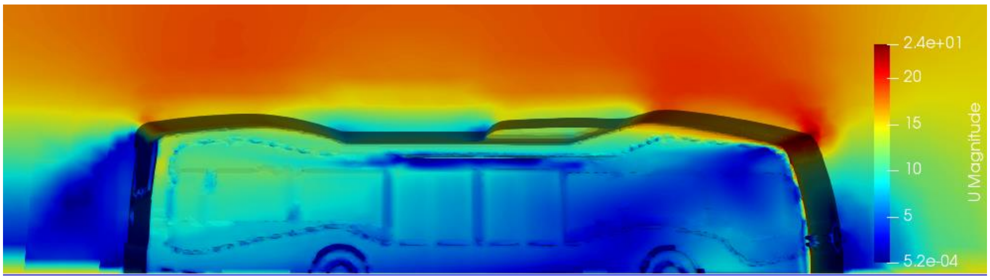
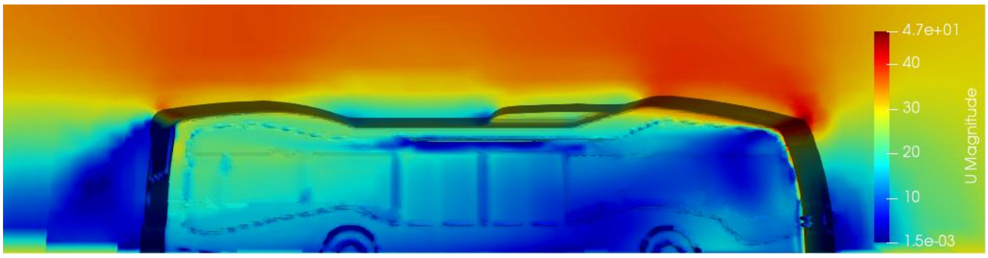
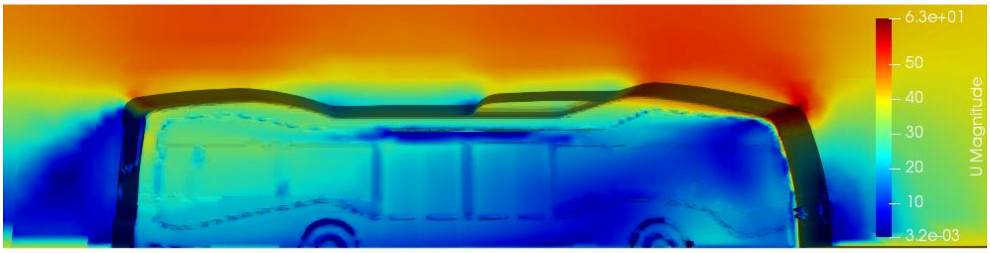
Inlet velocity (m/s)	**Velocity contour**
15	
30	
40	

Figure 6: Comparison of velocity contour at different inlet velocities.

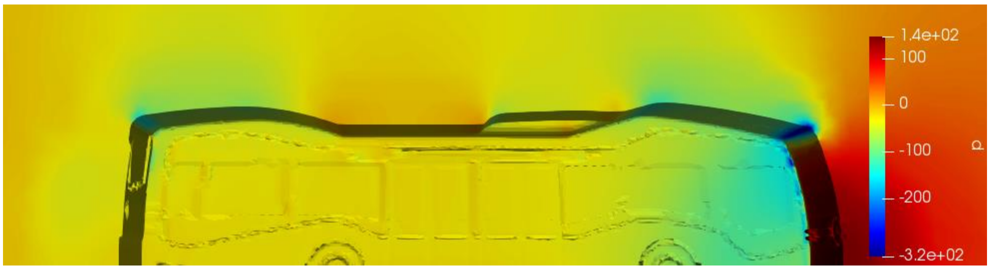
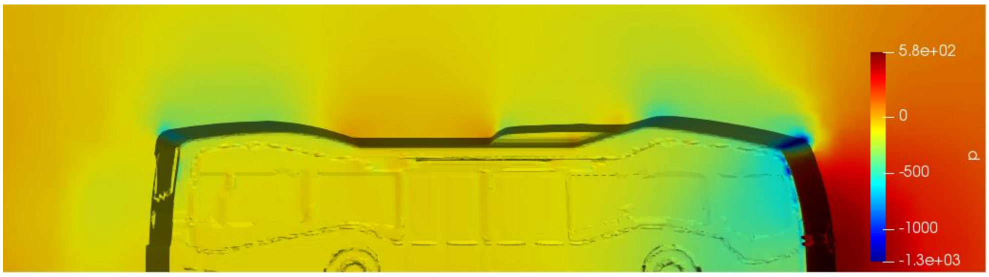
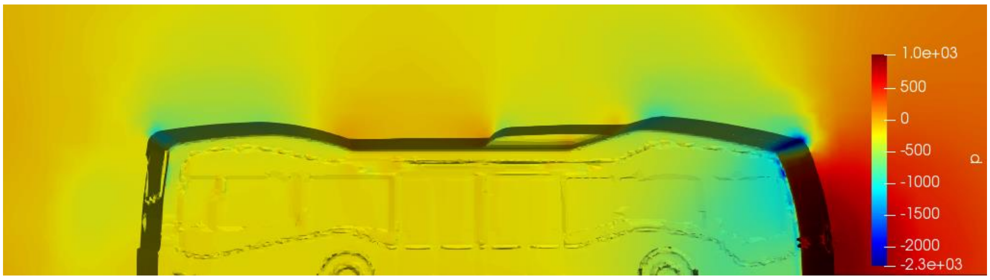
Inlet velocity (m/s)	Pressure contour
15	
30	
40	

Figure 7: Comparison of pressure contour at different inlet velocities.

The observations from the analysis reveal exciting phenomena regarding airflow dynamics and pressure distribution around the bus under specific inlet velocity conditions. At an inlet velocity of 5 m/s, the maximum velocity recorded was 7.8 m/s. Notably, areas of high velocity were concentrated at the top of the bus, as indicated by the red regions. Conversely, the lower front and rear of the bus experienced the weakest velocities, registering nearly 0 m/s. Regarding pressure distribution, the highest pressure recorded was 16 Pa, with the red region at the front of the bus indicating high-pressure areas. Much of the surrounding area exhibited a pressure of 0 Pa, depicted by the yellow regions. However, at specific points, such as the tip of the upper front, blue regions indicated negative pressure, valued at -36 Pa. Furthermore, it's noteworthy that both turbulent models displayed almost identical velocity and pressure contours, as depicted in Figures 8 and 9, respectively. These findings provide valuable insights into the complex interplay between airflow patterns and pressure distribution around the bus, shedding light on the effectiveness and consistency of the turbulent models utilized in the analysis.

The findings from the analysis of airflow dynamics and pressure distribution around the bus offer significant advantages for engineers in bus design. By pinpointing areas of high and low velocity and pressure distribution, engineers can optimize the bus design to minimize aerodynamic drag, improving fuel efficiency and overall performance. Understanding pressure distribution aids in reinforcing the structural integrity of the bus, enhancing safety and durability. Detailed velocity and pressure contours also inform the tailored design of specific components, such as spoilers and side skirts, further reducing drag and increasing efficiency [15]. Moreover, the consistency observed in turbulent models validates their accuracy, enabling more confident decision-making during the design process. Leveraging computational simulations reduces the need for costly physical testing, accelerates design iterations, and ultimately saves time and money. These insights advance bus design, optimizing performance and efficiency while enhancing safety and reducing environmental impact.

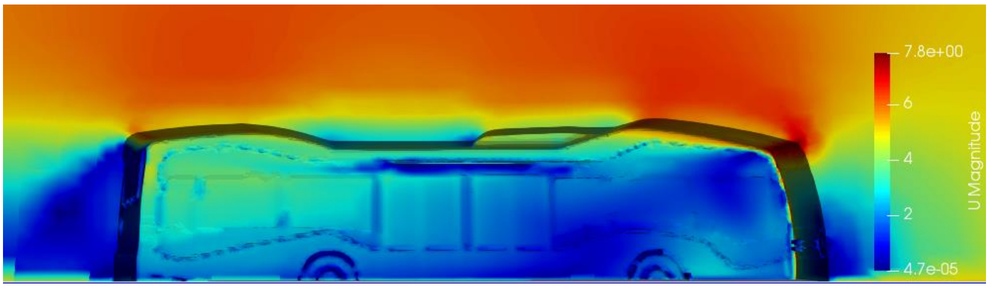
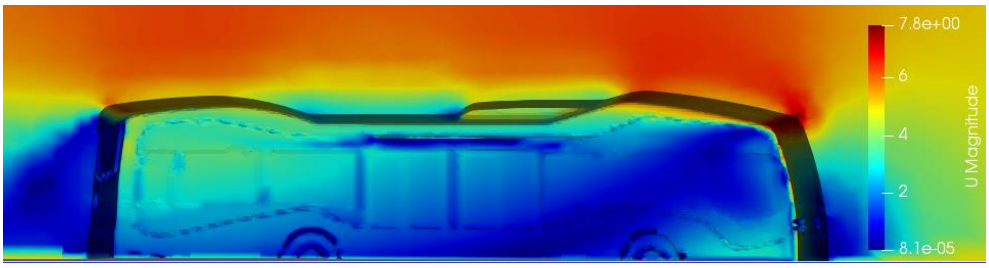
Model	Velocity contour
k-ω SST	
k-ε	

Figure 8: Comparison of velocity contour for different turbulent models at 5m/s of inlet velocity.

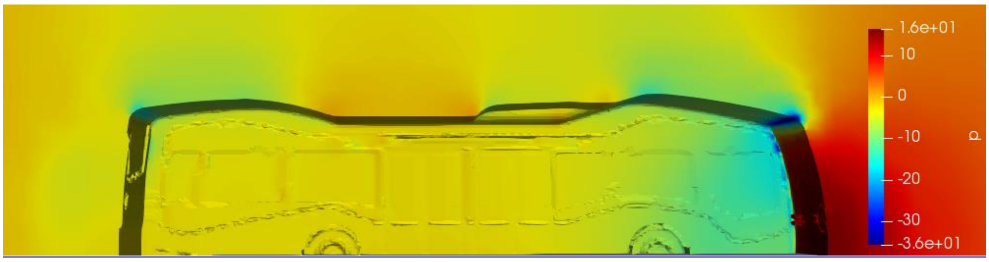
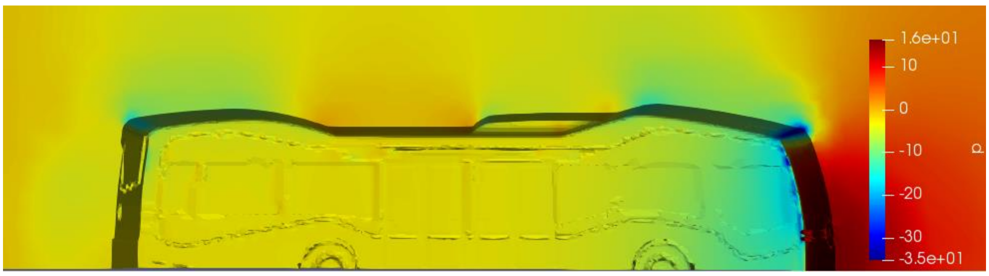
Model	Pressure contour
k-ω SST	
k-ε	

Figure 9: Comparison of velocity contour for different turbulent models at 5m/s of inlet velocity.

4. CONCLUSION

Studying the impact of airflow velocities around buses through Computational Fluid Dynamics (CFD) analysis proves pivotal in addressing engineering challenges. This study successfully simulated and examined a bus body using CFD post-processing, elucidating velocity and pressure contours to deepen our understanding of its aerodynamic behavior. The findings revealed a direct proportional relationship between inlet velocity and maximum velocity, highlighting the influence of higher inlet velocities on achieving greater maximum velocities. Additionally, a strong correlation between inlet velocity and pressure underscored the significance of inlet velocity in determining pressure distribution. While differences in turbulent models showed limited impact at low velocities, further investigation under varied flow conditions could enhance large vehicle design, bolstering resilience against storms and improving fuel efficiency. Enhancing bus design optimizes fuel consumption and promotes energy-efficient transportation from point A to point B, highlighting the importance of ongoing research and refinement in pursuing high-performance bus designs.

ACKNOWLEDGEMENTS

The author thanks the support provided by the Faculty of Mechanical Engineering & Technology throughout this research.

REFERENCES

[1] Kaushik, M. Theoretical and experimental aerodynamics Singapore: Springer. (2019) pp. 107-126.

[2] Hucho, W. H. (Ed.). Aerodynamics of road vehicles: from fluid mechanics to vehicle engineering. Elsevier. (2013).

[3] Lewin, T. Speed Read Car Design: The History, Principles and Concepts Behind Modern Car Design. (2017).

[4] Sheth, C., Triantis, K., & Teodorović, D. Performance evaluation of bus routes: A provider and passenger perspective. Transportation Research Part E: Logistics and Transportation Review, vol 43, issue 4 (2007) pp. 453-478.

[5] Sudin, M. N., Abdullah, M. A., Shamsuddin, S. A., Ramli, F. R., & Tahir, M. M. Review of research on vehicles aerodynamic drag reduction methods. International Journal of Mechanical and Mechatronics Engineering, vol 14, issue 2 (2014) pp. 37-47.

[6] Mohamed, E. A., Radhwi, M. N., & Abdel Gawad, A. F. Computational investigation of aerodynamic characteristics and drag reduction of a bus model. American Journal of Aerospace Engineering, vol 2, issue 1 (2015) pp. 64-73.

[7] Sivaraj, G., Parammasivam, K. M., & Suganya, G. Reduction of aerodynamic drag force for reducing fuel consumption in road vehicle using basebleed. Journal of Applied fluid mechanics, vol 11, issue 6 (2018) pp. 1489-1495.

[8] Raveendran, A., Sridhara, S. N., Rakesh, D., & Shankapal, S. R. Exterior styling of an intercity transport bus for improved Aerodynamic performance (2009)(No. 2009-28-0060)

[9] 9. Jadhav, C. R., & Chorage, R. P. Modification in commercial bus model to overcome aerodynamic drag effect by using CFD analysis. Results in Engineering, vol 6, (2020) p. 100091.

[10] 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. Drag reduction of a commercial bus with add-on aerodynamic devices. Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering, vol 237, issue 7 (2023) pp. 1623-1636.

[11] Abinesh, J., & Arunkumar, J. CFD analysis of aerodynamic drag reduction and improve fuel economy. International Journal of Mechanical Engineering and Robotics Research, vol 3, issue 4 (2014) p. 430.
[12] Kanekar, S., Thakre, P., & Rajkumar, E. Aerodynamic study of state transport bus using computational fluid dynamics. In IOP Conference Series: Materials Science and Engineering, vol 263, issue 6, (2017) p. 062052
[13] Sadadiwala, V., & Sharma, A. C. D. A Detailed Study: CFD Analysis of NACA 0012 at Varying Angles of Attack. International Journal for Research in Applied Science and Engineering Technology, vol 9, (2021) pp. 2330-2336.
[14] Tian, L., Li, Y., Li, J., & Lv, W. A simulation based large bus side slip and rollover threshold study in slope-curve section under adverse weathers. Plos one, vol 16, issue 8 (2021) p. 0256354.
[15] Hyams, D. G., Sreenivas, K., Pankajakshan, R., Nichols, D. S., Briley, W. R., & Whitfield, D. L. Computational simulation of model and full scale Class 8 trucks with drag reduction devices. Computers & Fluids, vol 41, issue 1, (2011) pp. 27-40.

Conflict of interest statement: The author declares no conflict of interest.

Author contributions statement: Conceptualization; Methodology; Software; Analysis; Investigation; Writing & Editing, M.A.H. Samsuddin.