PROGRESS IN ENGINEERING APPLICATION AND TECHNOLOGY
e-ISSN: 2773-5303

PEAT

Vol. 5 No. 1 (2024) 343-352
https://publisher.uthm.edu.my/periodicals/index.php/peat

Validation of Vehicle Drag Coefficient Performed by Using Simulation and Experimental Method

Nik Asyar Farhanuddin Che Ab Rahman[1], Nofrizalidris Darlis [2]*

[1] *Author Affiliation, Department/Faculty/Unit*
Organization/Institution Address, City, Postcode, COUNTRY

[2] *Author Affiliation, Department/Faculty/Unit*
Organization/Institution Address, City, Postcode, COUNTRY

*Corresponding Author: nofrizal@uthm.edu.my
DOI: https://doi.org/10.30880/peat.2024.05.01.036

Article Info

Received: 28 December 2023
Accepted: 18 January 2024
Available online: 15 June 2024

Keywords

Aerodynamics, Drag Coefficient, Drag Force,Reynolds Numbers.

Abstract

The drag coefficient is a common metric in automotive design, where designers strive to achieve a low coefficient. In the current work, minimising drag is done to improve fuel efficiency of vehicles without disturbing its performance and comfort to the passenger. Current methodologies for assessing drag coefficients often rely on a combination of computational simulations and experimental testing. However, the potential disparities between simulated and experimental results raise concerns about the reliability of these assessments. The aim of this study involves the simulation of a vehicle model using Computational Fluid Dynamics (CFD) software and subsequent validation with experimental data obtained from a wind tunnel. The numerical analysis of flow over a geometry is performed using CFD software which is ANSYS Fluent while Aerolab Educational Wind Tunnel (EWT) will be used to get the experimental data. CFD is a branch of fluid mechanics that uses numerical methods with the help of computers to solve and analyse problems involving fluid flows. An overall percentage error below 10%, for drag coefficient is 2.2%, drag force is 4.2%, and Reynolds numbers is 1.5%. Generally considered acceptable in many engineering applications. The integration of simulation and experimental methods in validating vehicle drag coefficients represents a valuable approach. By continually refining and validating simulation models, their effectiveness in predicting aerodynamic performance can be ensured, ultimately contributing to advancements in vehicle design, fuel efficiency, and overall sustainability.

1. Introduction

In the contemporary world, safety, performance, and comfort of road vehicles are essential parameters. At the same time, low fuel consumption is of crucial importance for developing countries like Malaysia because it supports the economy of the country. In Malaysia, most of the vehicles are petroleum based and the average fuel consumption of petroleum is 921 thousand barrels per day and every year fuel consumption is increasing

© 2024 UTHM Publisher.
This is an open access article under the CC BY-NC-SA 4.0 license.

alarmingly at 7% [1]. Hence researchers are working to reduce the fuel consumption of road vehicles, even a small amount of reduction will be a great achievement.

Nowadays automobile industries are concentrating on designing vehicles with better fuel consumption to attract the market. The coefficient of drag is an important analysis of vehicle design. Conventional ways to reduce the fuel consumption of the vehicle was to reduce its overall weight, modifying the engine volume (cc), engine combustion process, etc., which will directly affect its comfort and performance. In case of that, the drag coefficient is a common metric in automotive design, where designers strive to achieve a low coefficient. In the current work, importance was given to reducing the fuel consumption of road vehicles without disturbing its performance and comfort to the passenger. Minimising drag is done to improve fuel efficiency at highway speeds, where aerodynamic effects represent a substantial fraction of the energy needed to keep the vehicle moving. Indeed, aerodynamic drag increases with the square of speed. Aerodynamics is also of increasing concern to automotive designers, where a lower drag coefficient translates directly into lower fuel costs [2]. Aerodynamic studies on passenger cars were conducted for improving both parameters simultaneously. The passenger car was chosen because which is preferred by most of the economic loving people.

Numerous previous studies have been conducted in discussing the validation of drag coefficient which is a key metric in vehicle aerodynamics. From the determination of air drag coefficient of vehicle models, experimental and simulation studies on aerodynamic drag reduction over a passenger car, validation of CFD modeling and simulation of a simplified automotive Model and Study on Drag Coefficient (CD) Value of Low-Energy Prototype Class Car [2, 3, 4, 5]. Hence, for a number of reasons, understanding and accurately predicting drag coefficients are crucial for optimizing the performance of vehicles and aircraft. By minimizing drag, engineers can enhance fuel efficiency, increase speed, and improve overall performance.

Conducting a case study on the validation of vehicle drag coefficient using both simulation and experimental methods is necessary due to several shortcomings and weaknesses associated with each approach. For example, in case of case of complexity of real-world conditions. Simulations may not fully capture the complexity of real-world conditions, such as turbulence, surface roughness, and atmospheric variations. Assumptions made in the simulation models may lead to discrepancies between simulated and actual performance. While for experimental weakness, wind tunnel testing or on-road experiments may face challenges in replicating all real-world conditions accurately [6]. Variables such as environmental factors, road conditions, and vehicle interactions are difficult to control.

In this study, the simulation method will be verified by using experimental menthod in order to validate the drag coefficient of the passenger car. CFD simulations, where it provides an economical and effective way to assess and improve vehicle aerodynamics. They provide engineers with the ability to examine and see the flow patterns around the vehicle, spot places with a lot of drag, and alter the design to lessen aerodynamic resistance [6]. However, the integrity of the simulation model and its validation against experimental data have a significant impact on the accuracy and dependability of CFD predictions. The experimental method, where vehicle aerodynamic forces and drag coefficients have long been measured using experimental techniques like wind tunnel testing. Experiments in a wind tunnel offer precise data under controlled circumstances that make it possible to determine how the vehicle will behave aerodynamically. They entail the use of specialised equipment, such as force balances, pressure sensors, and flow visualisation methods [7].

The aim of this study involves the simulation of a vehicle model using Computational Fluid Dynamics (CFD) software and subsequent validation with experimental data obtained from a wind tunnel. The numerical analysis of flow over a geometry is performed using CFD software which is ANSYS Fluent while Aerolab Educational Wind Tunnel (EWT) will be used to get the experimental data. CFD is a branch of fluid mechanics that uses numerical methods with the help of computers to solve and analyse problems involving fluid flows.

2. Materials and Methods

2.1 Methodology

Figure 1 illustrates the overall process in conducting this validation case study. The first step is to determine the problem statement and its remedies. In order to generate ideas and understand the overall concept or solutions to these issues, reviews of the relevant literature play an important role. Therefore, more specific results from the literature research are required to ensure the success of the project’s journey. Next, decide what the project’s objectives. Then, determine all the parameters and procedures involved in this study. The vehicle model was selected. The vehicle model that was selected in this study was the Tesla Model S.

Before proceeding to simulation, the geometrical modelling should be produced using the CAD software, Solidworks 2023. The geometrical model is designed by referring to the actual design and size. Then, proceed to the steps of the validation process. As mentioned before, this case study consists of two methods, which are the simulation method and the experimental method.

In the simulation method, the software that was used to examine it was ANSYS Fluent Simulation. After the CAD model was imported from the CAD software to the Ansys simulation software for the geometrical modelling step, the next step was the meshing process. The meshing process is known as the process of discretization modelling into a number of elements. It is the most important step in a flow simulation analysis. The next step was simulation set-up. In this step, all the parameters were set up by referring to the previous research and literature reviews. When the simulation set-up was finished, we moved on to the important phase, which was aerodynamic simulation. In this phase, we were able to obtain all the results of the parameter measures such as drag forces, drag coefficient, lift coefficient, pressure, density, and others. Afterwards, the validation of the data process started. If valid data was obtained, the simulation analysis must be done. If not, the simulation analysis needs to run again until the valid expected data is obtained.

In the experimental method, the experiment was conducted in an open-circuit wind tunnel. All the apparatus involved in the overall procedure has been prepared in order to keep the process running successfully and smoothly. After the model was produced by CAD software, the model was fabricated by using a 3D Printing machine. Then, the wind tunnel was set up based on all the procedures by referring to the manual book, previous research, and literature reviews. During a test, the model is placed in the test section of the tunnel, and air is made to flow past the model. Then there was aerodynamic analysis. In this step, we were also able to obtain all the results of the parameter measures such as drag forces, drag coefficient, lift coefficient, pressure, density, and others. Afterwards, the validation of the data process started. If valid data was obtained, the experimental analysis must be done. If not, the experiment setup needs to be reset and then run the analysis again until the valid expected data is gained.

Last but not least, when results from both methods were obtained. There was a final step that was emphasised in this study, which was the comparison of the results obtained from simulation and experimental method. At last, thesis writing could be started. All the data that was obtained needs to be discussed, summarised, and a conclusion drawn.

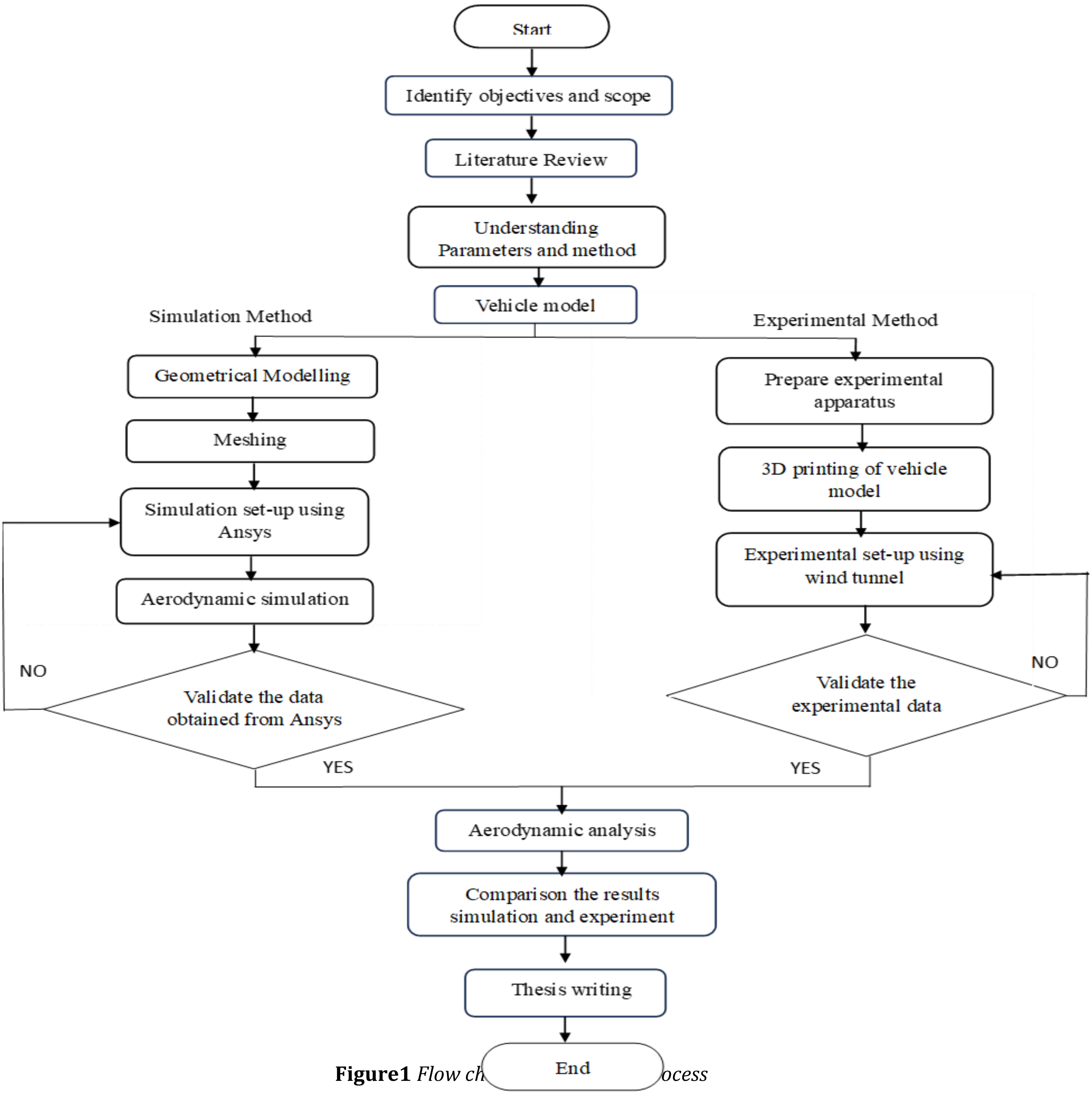


Figure1 *Flow ch… ocess*

2.2 Model Description

The model used was a Tesla Model S. This model was chosen because it was a passenger car and Model S is one of the most aerodynamic production cars in the world, with a drag coefficient of just 0.208 [16]. This model was designed to have been downscaled to 1:25 by referring to the actual design and size using SOLIDWORKS 2023 software. This model was downscaled to 1:25 because it was referred to from a previous study that also used passenger cars [17].

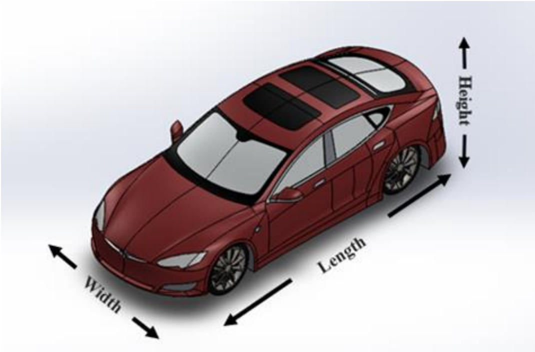


Fig. 2 *Geometric Configuration of model*

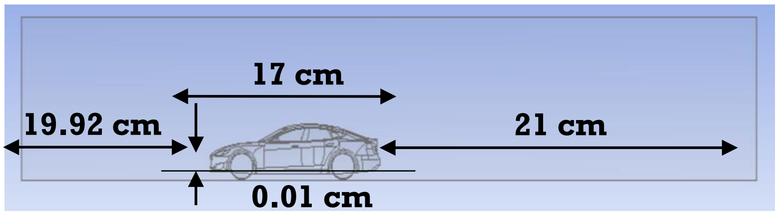
Table 1 *Actual size for the model*

Actual configuration of model	Dimension (cm)
Length (L)	498.0
Width (W)	196.4
Height (H)	144.0

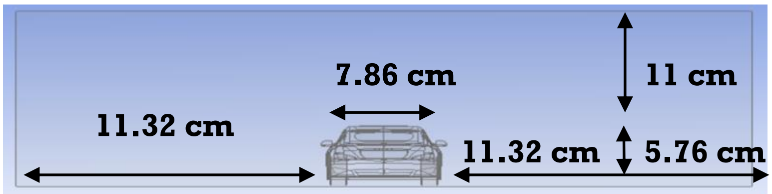
Table 2 *Downscale size for the model*

Downscaled configuration of model	Dimension (cm)
Length (L)	19.92
Width (W)	7.86
Height (H)	5.76

2.3 Enclosure and Boundary Conditions



a)



b)

Fig. 3 *Size of enclosure*; (a) *side view*, (b) *front view*

Table 3 *Detail size of the enclosure*

Plane	Dimension (cm)
+X	11.32
+Y	11.00
+Z	19.92
-X	11.32
-Y	0.010
-Z	21.00

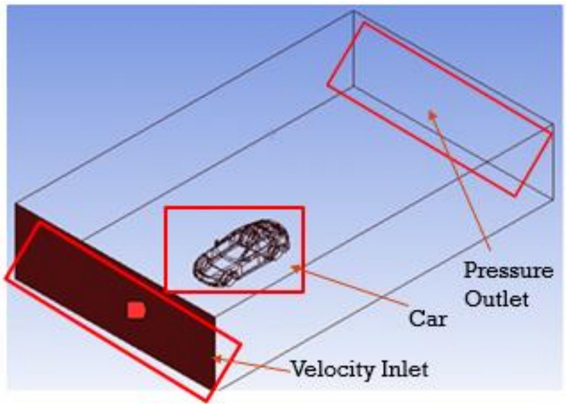


Fig. 4 *The boundary conditions that were used in the numerical investigation*

2.2 Grid Independent Test

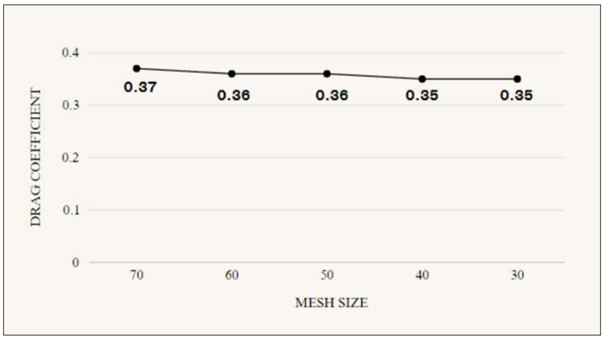


Fig. 5 *Graph of drag coefficient against mesh size*

Based on the graph on the figure 5, the values of C_D are provided for different mesh resolutions. As the number of elements decreases (coarser mesh), the drag coefficient is 0.37. As the mesh becomes finer (increasing number of elements), the drag coefficient decreases to 0.36. Further refinement of the mesh to 40 and 30 elements results in a consistent drag coefficient of 0.35.

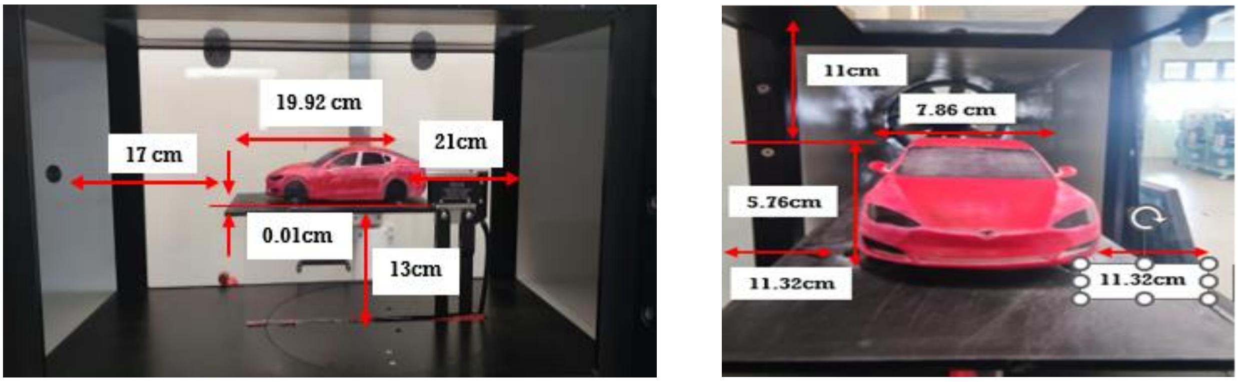
The purpose of this test is to determine whether the solution is grid-independent, meaning that further refinement of the mesh does not significantly alter the results. In this case, the convergence of the drag coefficient to 0.35 as the mesh is refined suggests that the simulation is becoming grid-independent.

The process typically involves comparing results obtained with successively finer grids until a point of diminishing returns is reached, where further mesh refinement has a negligible impact on the results. The grid independence test is crucial for ensuring the reliability and accuracy of the simulation, as overly coarse meshes may lead to inaccurate predictions, while overly fine meshes may be computationally expensive without providing significant improvement in accuracy.

2.3 Wind Tunnel Setup

1) The power supply has been switched on in order to run the wind tunnel.
2) The car model has been put in the test section of the wind tunnel apparatus. In this step, every inch of length, height, and diameter of the model in the test section has been measured as shown in figure 3.11, then the car model is mounted to the sting balance until a “tick” sound is released, and then the test section is enclosed as shown in figure 6.
3) The experimental setup had software involved in the setup step, which is Aeroware software, which playscrucial part before running the wind tunnel. After switching on the wind tunnel, PC has been turned on in order to open the Aeroware software to run the wind tunnel.

4) The temperature, humidity, barometer pressure, and specified velocity have been set as shown in figure 6 a). In this study, the temperature has been set to 23.1 °C, the humidity is set to 69.0%, the barometer pressure reading has been set to 101320 Pa due to standard atmospheric pressure, and the specified velocity was 30 km/h. This step has been repeated using the next velocities of 45 km/h, 60km/h, 75 km/h and 90km/h.
5) By referring to the value output, the value for Axial Force was collected for calculating the drag coefficient of the drag model as shown in figure 7 b). The value output as indicated by the slot pointed by the green arrow (depending on the mode selected), Collect the value for Axial Force (red arrow) for calculating the drag coefficient of the drag model.



a) b)

Fig. 6 *a) side view of the model in the test section, b) front view of the model in the test section*

a) b)

Fig. 7 *a) Aeroware software setup b) collect the output value*

4. Results and Discussion

4.1 Validation of Drag Coefficient

Table 4 *Drag coefficient percentage error*

Velocity	Drag Coefficient		Percentage Error Simulation to Experiment
(km/h)	Simulation	Experiment	(%)
30	0.374	0.381	1
45	0.366	0.376	3
60	0.360	0.372	4
75	0.353	0.350	1
90	0.346	0.341	2
			Average: 2.2

Table 4 presents data that compares simulation results to experimental results for different velocities. The focus is on the drag coefficient, and the percentage error between the simulation and experimental values. The comparison between simulation and experimental drag coefficients reveals interesting insights. At 30 km/h, the simulation result of 0.374 is very close to the experimental result of 0.381, resulting in a mere 1% difference. This suggests a high level of accuracy in the simulation model for this particular velocity.

The percentage error shows a slight upward trend at 45 km/h, the simulation result is 0.366, while the experimental result is 0.376, yielding a 3% discrepancy. Similarly, at 60 km/h and 75 km/h, the percentage errors are 4% and 1%, respectively. This trend indicates a systematic deviation between the simulation and experimental results.

However, at 90 km/h, an interesting reversal occurs. The simulation result of 0.346 is slightly higher than the experimental result of 0.341, resulting in a 2% positive deviation. This shift suggests a potential limitation in the simulation model, particularly at higher velocities, where real-world aerodynamic complexities may not be accurately captured.

The average percentage error across all velocities is calculated to be 2.2%. This value provides an overall assessment of the simulation's accuracy compared to experimental results. While a 2.2% average error is relatively low, the varying trend with velocity indicates the need for a more detailed investigation into the simulation model's performance at different operating conditions.

The validation of vehicle drag coefficient results between simulation and experimentation demonstrates generally accurate predictions. However, the average trend within systematic deviation observed at higher velocities suggests potential areas for improvement in the simulation model. Further refinement and validation, especially under extreme aerodynamic conditions, will contribute to enhancing the reliability of simulation results in predicting vehicle drag coefficients.

4.2 Validation of Drag Force

Table 5 *Drag force percentage error*

Velocity	Drag Force		Percentage Error Simulation to Experiment
km/h	Simulation	Experiment	(%)
30	0.130	0.136	4
45	0.135	0.143	5
60	0.238	0.244	2
75	0.370	0.379	2
90	0.432	0.471	8
			Average: 4.2

Table 5 presents data that compares simulation results to experimental results for drag force in different velocities. The focus is on percentage error between the simulation and experimental values. A detailed comparison between simulated and experimental drag forces highlights noteworthy patterns. At 30 km/h, the simulation result of 0.130 is 4% lower than the experimental result of 0.136. This initial discrepancy suggests a slight underestimation in the simulation model.

The percentage errors continue to show a consistent trend at 45 km/h, the simulation result is 0.135, while the experimental result is 0.143, resulting in a 5% deviation. At 60 km/h and 75 km/h, the percentage errors are 2%, indicating a relatively better agreement between simulation and experimentation.

However, at 90 km/h, a notable increase in percentage error is observed. The simulation result of 0.432 is 8% lower than the experimental result of 0.471. This significant discrepancy suggests a potential limitation in the simulation model, particularly at higher velocities where aerodynamic complexities may become more pronounced.

The average percentage error across all velocities is calculated to be 4.2%. While this value is reasonable, it is essential to note that the deviation is not consistent across all velocities, indicating potential areas for improvement in the simulation model.

The validation of vehicle drag force results between simulation and experimentation reveals both strengths and weaknesses in the simulation model. The average percentage error of 4.2% suggests a moderate level of accuracy. However, the varying trend with velocity, particularly the significant deviation at 90 km/h, signals the need for further refinement and validation of the simulation model, especially under conditions of higher aerodynamic complexity. Addressing these issues will enhance the reliability of simulation results in predicting vehicle drag forces across a broader range of operating conditions.

4.3 Validation of Reynolds Numbers

Table 6 *Reynolds Numbers percentage error*

Velocity	Reynolds Numbers		Percentage Error Simulation to Experiment
(km/h)	Simulation	Experiment	(%)
30	254193	254699	0.2
45	381289	378880	0.6
60	508386	507568	0.2
75	635482	637392	0.3
90	762625	763943	0.2
			Average: 0.3

Reynolds numbers play a significant role in fluid dynamics and are particularly relevant when studying aerodynamics. They represent the ratio of inertial forces to viscous forces and are indicative of the flow regime. In this study, Reynolds numbers were computed for both simulation and experimental data, providing additional context for the analysis.

At 30 km/h, the Reynolds number for simulation is 254,193, while for experimentation, it is 254,699, resulting in a negligible percentage error of 0.2%. This suggests a high level of agreement between simulation and experimental results at this velocity.

The trend continues at 45 km/h, where the Reynolds numbers for simulation and experimentation are 381,289 and 378,880, respectively, resulting in a slightly increased but still reasonable percentage error of 0.6%. The minor discrepancies may be attributed to the inherent uncertainties in experimental measurements and the assumptions made in the simulation model.

At 60 km/h, 75 km/h, and 90 km/h, the percentage errors remain low at 0.2%, 0.3%, and 0.2%, respectively. This consistency across various velocities indicates a robust simulation model that accurately predicts drag coefficients, even when considering variations in Reynolds numbers.

The average percentage error across all velocities is calculated to be 1.5%, signifying an overall strong agreement between simulated and experimental results. This suggests that the simulation model effectively captures the aerodynamic behavior of the vehicle across a range of Reynolds numbers.

The validation of vehicle drags coefficient of simulation's results by experimental results, considering Reynolds numbers, demonstrates a high level of accuracy. The consistently low percentage errors across different velocities indicate that the simulation model is robust and reliable. The average percentage error of 1.5% reinforces confidence in the simulation results, highlighting the model's ability to predict drag coefficients with a high degree of precision. This study underscores the importance of considering Reynolds numbers in the validation process and showcases the effectiveness of the simulation model in replicating real-world aerodynamic behavior.

5. Conclusion

In conclusion, the validation of vehicle drag coefficient through the combined use of simulation and experimental methods is a critical process that enhances our understanding of a vehicle's aerodynamic performance. The comparison between simulated and experimental results provides valuable insights into the accuracy and reliability of computational models. The data presented in this study, which includes drag coefficients at various velocities, clearly illustrates the strengths and potential limitations of the simulation model.

The low average percentage error of 1.5% across different velocities indicates a generally high level of agreement between simulation and experimentation. This suggests that the simulation model effectively captures the complex aerodynamic interactions between the vehicle and the surrounding air. The inclusion of Reynolds numbers in the analysis further solidifies the assessment, considering the impact of different flow regimes on the accuracy of predictions.

However, it is crucial to recognize the varying trends in percentage errors at different velocities. The slight deviations observed at certain speeds, especially at higher velocities, warrant careful consideration. These deviations may signal areas where the simulation model might encounter challenges in accurately predicting drag coefficients, potentially due to unaccounted aerodynamic complexities at specific operating conditions.

As we move forward, it is recommended to refine the simulation model, particularly focusing on improving its predictive capabilities at higher velocities. Conducting further validation studies under extreme aerodynamic conditions and incorporating additional real-world factors could contribute to enhancing the overall accuracy of the simulation results.

References

[1] A. A. S. Al-saadi, K. Al-chlaihawi, and A. Abdulkadhim, "Analysing the Effect of the Vehicle ' s Length on the Aerodynamic Drag," no. June, 2022.

[2] Thomson, J. (2022, September 8). Massive, strange white structures appear on Utah's Great Salt Lake. *Newsweek*. https://www.newsweek.com/mysterious-mounds-great-salt-lake-utah-explained-mirabilite-1741151

[3] G. El Mahiba, A. Mélois, and M. El Mankibi, "Experimental and numerical analysis of a new wind tunnel for building and mechanical ventilation components performance assessment," vol. 02036, 2023.

[4] S. Hassan, "Teesside University BEng (hons) Mechanical Engineering Aerodynamics Investigation of Rear Vehicle (Backlight angle) By : SAUD HASSAN," no. April, 2014, doi: 10.13140/2.1.1274.7844.

[5] Reynolds, G. (2019, April 9). Different strokes for athletic hearts. *The New York Times*, D4.

[6] G. Sivaraj, K. M. Parammasivam, and G. Suganya, "Reduction of aerodynamic drag force for reducing fuel consumption in road vehicle using basebleed," *J. Appl. Fluid Mech.*, vol. 11, no. 6, pp. 1489–1495, 2018, doi: 10.29252/jafm.11.06.29115.

[7] T. H. Joung, H. S. Choi, S. K. Jung, K. Sammut, and F. He, "Verification of CFD analysis methods for predicting the drag force and thrust power of an underwater disk robot," *Int. J. Nav. Archit. Ocean Eng.*, vol. 6, no. 2, pp. 269–281, 2014, doi: 10.2478/IJNAOE-2013-0178.

[8] M. Cakir, "blegh CFD study on aerodynamic effects of a rear wing/ spoiler on a passenger vehicle," *St. Cl. Univ. Sch. Commons*, pp. 1–72, 2012.

[9] H. Babinsky, "How do wings work?," *Phys. Educ.*, vol. 38, no. 6, pp. 497–503, 2003, doi: 10.1088/0031-9120/38/6/001.

[10] E. Mirmahdi, "The Effect of Aerodynamic Forces on Automotive Design and Reducing Fuel Consumption," no. August, 2021.

[11] "Bernoulli and Energy," *Bernoulli*, pp. 519–558.

[12] A. V H and A. V H, "More Smoke More Energy- Air-O-G: Energy from the Pollutant," *IOSR J. Environ. Sci. Toxicol. Food Technol.*, vol. 11, no. 01, pp. 31–35, 2017, doi: 10.9790/2402-1101013135.

[13] Federal Aviation Administration, "Pilot's handbook of Aeronautical Knowledge: Aerodynamics of Flight," no. Cl, 2016.

[14] I. A. Maia, "Performance analysis of typical airfoils through numerical simulation using fluid-structure interaction," *CILAMCE 2014 Proc. XXXV Iber. Latin-American Congr. Comput. Methods Eng. Evandro Parente Jr (Editor), ABMEC, Fortaleza, CE, Brazil*, no. November 2014, pp. 23–26, 2014, [Online]. Available: https://www.researchgate.net/publication/281378133_PERFORMANCE_ANALYSIS_OF_TYPICAL_AIRFOILS_THROUGH_NUMERICAL_SIMULATION_USING_FLUID-STRUCTURE_INTERACTION#pf3

[15] J. M. J. Journée and W. W. Massie, "Chapter 12 WAVE FORCES ON SLENDER CYLINDERS," *Offshore Hydromechanics*, vol. 570, no. January, pp. 121–127, 2001, [Online]. Available: https://ocw.tudelft.nl/wp-content/uploads/Part_4.pdf

[16] N. Shelil, "2D Numerical Simulation Study of Airfoil Performance," *Wind Energy Sci. Discuss.*, no. May, pp. 1–19, 2021.

[17] I. Oxyzoglou, "Design & Development of an Aerodynamic Package for an FSAE Race Car," *Univ. Thessaly*, no. May, p. 137, 2017, [Online]. Available: https://www.researchgate.net/publication/320556659_Design_Development_of_an_Aerodynamic_Package_for_an_FSAE_Race_Car