Advanced and Sustainable Technologies
ASET
E-ISSN 2976-2294

Volume 3, No 1, June 2024 [54-61]

Effects of Race Car's Speed on the Aerodynamic Aspect Using Computational Fluid Dynamics Analysis

Yik Pey Tang[1*]

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

Received 5 April 2024, Revised 29 April 2024, Accepted 6 May 2024

ABSTRACT

This research employs Computational Fluid Dynamics (CFD) methods to investigate the intricate relationship between race car speed and external aerodynamics during high-performance racing competitions. The primary objectives encompass the application of CFD in pre-processing and analyzing external aerodynamic aspects, coupled with a comprehensive examination of the external flow around a race car for a nuanced understanding of its aerodynamic performance. Various car speeds were considered with the RANS (k-ω SST) turbulent model. The results unveiled a direct correlation between inlet velocity and the maximum velocity attained by the race car. The aerodynamic design intricately directs the airflow, leading to higher velocities predominantly along the upper part of the car body. Noteworthy is the revelation that the highest recorded maximum velocity of 231.06 m/s coincides with a peak inlet velocity of 200 m/s, suggesting a consistent increase in maximum velocity with rising inlet velocity. This research emphasizes the pivotal role of inlet velocity in achieving peak car speed performance. It sheds light on the significance of turbulent model selection in capturing the complexities of external flow dynamics. This knowledge contributes to optimizing the external aerodynamics of race car body design, ultimately enhancing performance and competitiveness in the dynamic world of Formula 1 racing.

Keywords: Computational fluid dynamics, Turbulent, Aerodynamic aspect, RANS.

1. INTRODUCTION

In the realm of Computational Fluid Dynamics (CFD) analysis, choosing the right solver is a critical factor, especially when working with diverse turbulence modeling strategies [1]. The selected solver significantly impacts the accuracy and computational efficiency of simulating turbulent flows. Just as meticulous meshing is essential, selecting the appropriate solver is crucial for ensuring reliable simulation outcomes. Various flow models, such as laminar, Reynolds-Averaged Navier-Stokes (RANS), and Large Eddy Simulation (LES) [2], require specific solver capabilities to effectively capture the complexities of turbulent flow behaviors. The choice of solver affects the convergence speed and stability of the simulation. It determines the level of detail and accuracy in capturing physical phenomena. For example, laminar flow models are generally more straightforward and require less computational power, making them suitable for flows with low Reynolds numbers where turbulence is minimal [3]. In contrast, RANS models, such as k-ε and k-ω, are designed to average the effects of turbulence, providing a balance between computational efficiency and accuracy for a wide range of engineering applications.

On the other hand, LES offers a more detailed approach by directly resolving large-scale turbulent structures while modeling more minor scales [4]. This method provides greater accuracy in

*Corresponding author: yipe12345@gmail.com

capturing transient and complex flow behaviors but comes at a higher computational cost. Consequently, the choice of solver must align with the specific requirements of the turbulence model employed, the nature of the flow being studied, and the available computational resources. The solver's ability to handle boundary conditions, such as inlet velocities, pressure outlets, and wall functions, is pivotal in accurately replicating real-world scenarios. Advanced solvers equipped with adaptive meshing techniques can dynamically refine the mesh in regions of high gradient, enhancing the precision of the simulation without excessively increasing computational demands [5]. Ultimately, the interplay between solver selection, turbulence modeling, and meshing defines the robustness and reliability of CFD simulations. Engineers must carefully consider these factors to ensure the simulated results are accurate and computationally feasible, enabling better prediction and analysis of fluid behaviors in complex systems.

Addressing the outlined problem in this module through CFD simulation involves four stages. The initial stage encompasses problem identification, defining modeling goals, and outlining the domain [6]. For instance, this project analyzes airflow within a car's domain. The second stage, pre-processing, involves creating a model, generating the mesh, and configuring the physics and solver settings [7]. This module specifically concentrates on assessing the airflow response by adjusting solver settings, including boundary conditions (such as inlet velocity) and turbulence setups (turbulence modeling). The subsequent stage is the solver phase, where computation and monitoring of the solution take place [8]. During this phase, the selected solver performs the necessary calculations to simulate the airflow, considering the defined boundary conditions and turbulence models. This stage is crucial for obtaining accurate and reliable results, as the solver must effectively capture the complex interactions within the airflow. The final stage, post-processing, involves scrutinizing the results and considering potential adjustments to the model based on the insights gained [9]. This includes analyzing the data to identify patterns, verifying the accuracy of the simulation against experimental or theoretical benchmarks, and making necessary refinements to improve the model. The goal is to ensure that the simulation results are both accurate and applicable to real-world scenarios, providing valuable insights into the airflow behavior within the car's domain.

The velocity of a racing car significantly affects its aerodynamic performance, playing a crucial role in determining the vehicle's stability, handling, and overall speed [10]. As the car's velocity increases, the aerodynamic forces acting on it, namely drag and downforce, become more pronounced. Higher velocities amplify the impact of these forces, necessitating precise aerodynamic design to minimize drag, which opposes forward motion, and to maximize downforce, which enhances tyre grip on the track. Effective management of these forces ensures that the car maintains optimal contact with the road, improving cornering speeds and reducing the risk of skidding or losing control [11]. Additionally, airflow patterns around the car become more turbulent at high velocities, requiring advanced computational fluid dynamics (CFD) simulations to accurately predict and optimize the aerodynamic characteristics [12]. Thus, understanding and optimizing the aerodynamic effects of varying car velocities is essential for achieving competitive performance in racing scenarios.

This paper discusses and compares the maximum values of velocity and pressure for varying car inlet velocities and turbulent model conditions. Firstly, the analysis explores the airflow response to car inlet velocities of 50 m/s, 100 m/s, and 200 m/s using the turbulent k-ω SST model. Additionally, the project examines the airflow response under different turbulent model conditions, including laminar, RANS (k-ε), and RANS (k-ω), with the car inlet velocity set at 50 m/s.

2. MATERIAL AND METHODS

The current simulation analysis was conducted using the SimFlow software [13]. The racing car model dimensions are 5.4 m × 1.30 m × 1.9 m. The 3-dimensional model of a racing car is shown in Figure 1. The flow domain used in the simulation was created at the minimum extent (-3, -0.2, 0.05) and maximum extent (15, 5, 5). The symmetrical domain is depicted in Figure 2. The domain meshing was created according to the number of divisions (50, 15, 20), and the cell size was 0.4 m for each axis. The dominant hexahedral elements were created in the meshing. After meshing, SimFlow showed the acceptable aspect ratio, maximum skewness, and non-orthogonal in the mesh quality checks. The current simulation uses the steady-state analysis, and the flow is assumed to be an incompressible flow condition. The SIMPLE solver [14] was selected in the SimFlow software, and the RANS k-ω SST turbulent model [15] was considered in the simulation. The boundary conditions were defined on the car model and the domain, with inlet velocity (X+), pressure outlet (X-), wall boundary for the car body, and bottom surface (Y-) of the domain. The symmetry plane was defined on the Z- surface, as shown in Figure 2. The simulation was converged at 200 iterations. Three inlet velocities (50, 100, and 200 m/s) were considered in the current analysis.

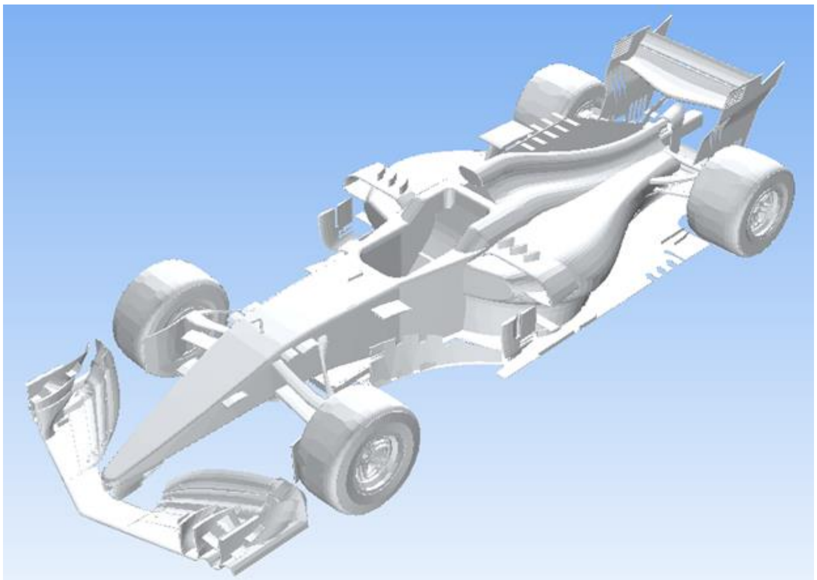


Figure 1: A racing car model is used in CFD analysis.

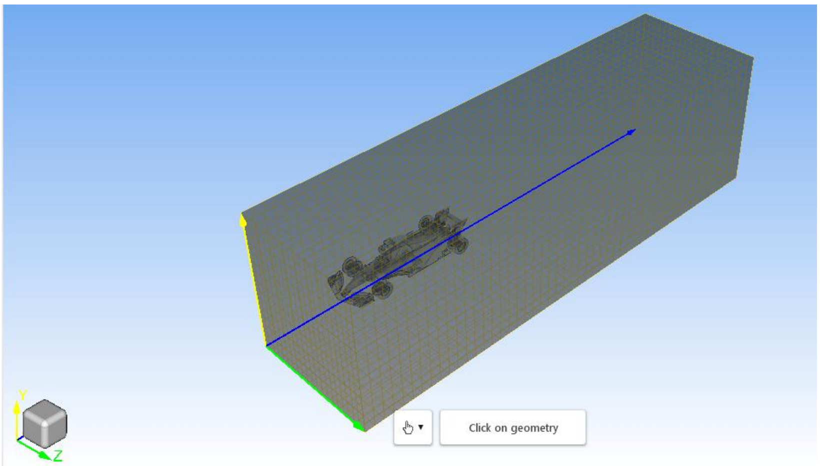


Figure 2: Symmetrical domain of a racing car.

3. RESULTS AND DISCUSSION

This section will present the results of a CFD analysis conducted on the car body. This section has shown the maximum velocity and maximum pressure recorded for each condition in terms of various inlet velocities and turbulent models. In addition, it has drafted three graphs and compared each change of result recorded. The following paragraph presents and discusses the maximum velocity and maximum pressure according to their color distribution diagram.

From Table 1, the highest recorded maximum velocity is 231.06 m/s. The color distribution diagram indicates that this peak velocity predominantly occurs along the upper part of the car body. This phenomenon suggests that the car's aerodynamic design directs airflow along the body and wings, accelerating air particles and resulting in higher velocities on the upper side. The graphical representation of the relationship between maximum velocity and inlet velocity illustrates a clear and direct correlation, showcasing a consistent increase as the inlet velocity rises. Notably, the highest maximum velocity coincides with the peak inlet velocity of 200 m/s. This correlation emphasizes the significant influence of inlet velocity on the maximum velocity attained by the car.

When examining the results for maximum pressure, we found that the highest recorded value was 13473 Pa. The color distribution diagram reveals that the maximum pressure is concentrated around the front and rear wings of the car body (Figure 3). This distribution suggests that air is redirected and forced through these wing structures, leading to increased pressure. Additionally, areas such as the engine cover or engine intake behind the driver also experience elevated pressure levels. The graphical representation of maximum pressure against inlet velocity indicates a direct relationship, characterized by a modest initial increase followed by a more substantial rise as the inlet velocity increases. Similar to the maximum velocity analysis, the peak maximum pressure aligns with the highest inlet velocity of 200 m/s, reinforcing the conclusion that changes significantly influence maximum pressure in inlet velocity.

Table 1: Turbulent model RANS (k-ω SST) with different inlet velocities.

Case	Inlet Velocity (m/s)	Maximum Velocity (m/s)	Maximum Pressure (Pa)
1	50	57.386	856.82
2	100	115.18	3433
3	200	231.06	13743

Table 2 summarizes the maximum velocity and pressure when using 50 m/s of inlet velocity. As shown in Figure 4, the highest recorded maximum velocity is 59.25 m/s, observed when utilizing the laminar flow model. This result indicates that the choice of turbulence model significantly impacts the simulation outcomes, specifically in terms of maximum velocity. The laminar model, which assumes smooth and predictable airflow, produces the highest velocity values. This underscores the importance of selecting an appropriate turbulence model when simulating airflow around the car. The varying characteristics of turbulence models reflect the airflow situations that may occur on the road.

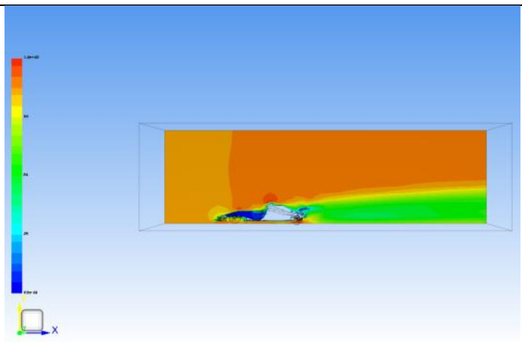
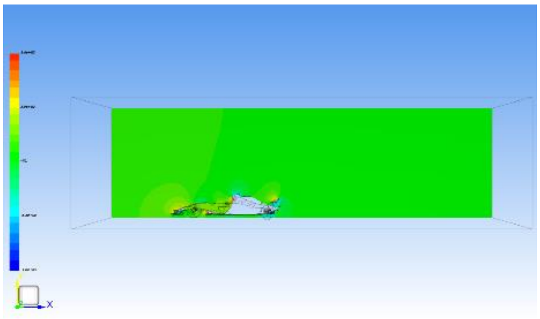
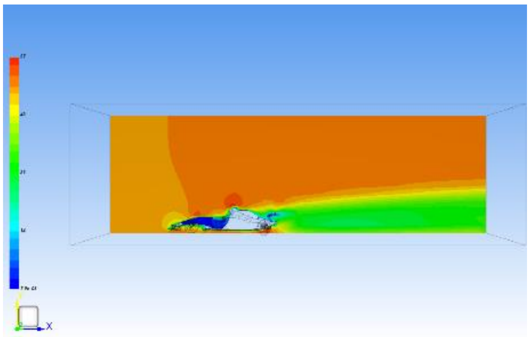
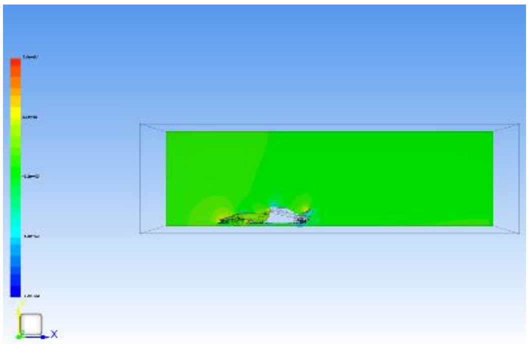
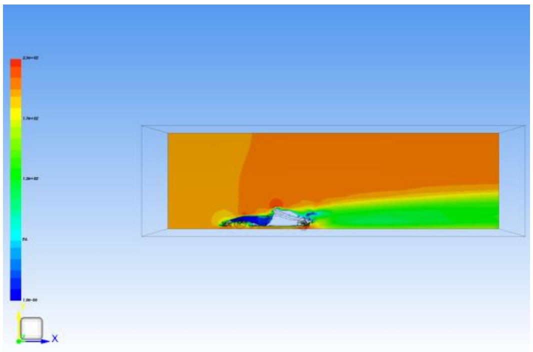
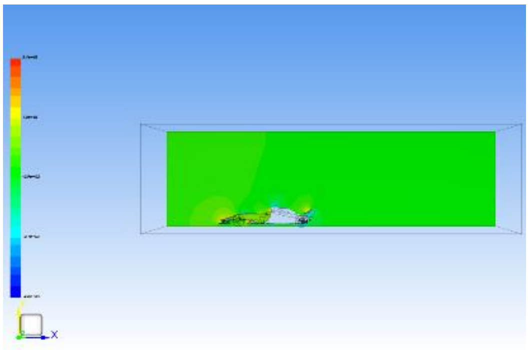
Inlet Velocity (m/s)	**Maximum Velocity (m/s)**	**Maximum Pressure (Pa)**
50		
100		
200		

Figure 3: Turbulent model as RANS (k-ω SST) with different inlet velocities.

Table 2: 50 m/s of inlet velocity with different flow models.

Case	**Flow Model**	**Maximum Velocity (m/s)**	**Maximum Pressure (Pa)**
1	Laminar	59.25	868.37
2	RANS (k- ε)	57.428	856.89
3	RANS (k-ω SST)	57.386	856.82

Similarly, the bar chart reveals that the highest maximum pressure value is 868.37 Pa, corresponding to the use of the laminar flow model. This finding emphasizes that the turbulence model employed plays a crucial role in determining the maximum pressure experienced by the car. The laminar model, which assumes a more orderly airflow, results in higher pressure values. This observation reinforces the notion that turbulence model selection directly influences the simulation outcomes and, consequently, the aerodynamic behavior of the F1 car.

In summary, the results lead to a conclusive insight: the values of maximum velocity and maximum pressure are dependent on the type of turbulence model used in the simulation. Each turbulence model is tailored to simulate specific airflow scenarios that may be encountered on the road. Therefore, when designing the car body, it becomes imperative to conduct simulations using various turbulence models to capture the diverse aerodynamic conditions that the F1 car may encounter during its racing endeavors. This approach ensures a comprehensive understanding of how different turbulent models influence the car's aerodynamic performance. It enables engineers to make informed design decisions to optimize its performance on the track.

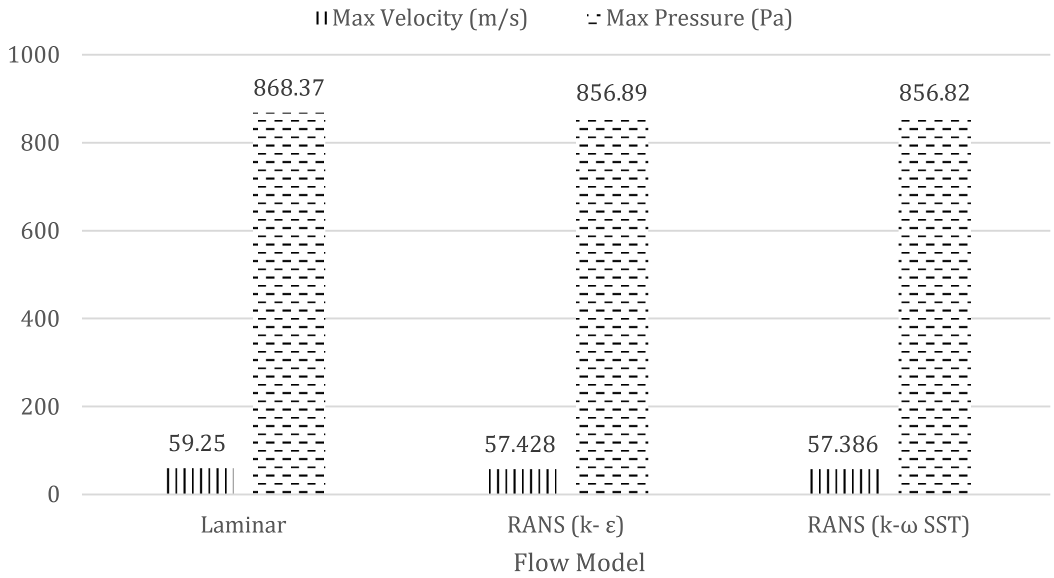


Figure 4: Maximum velocity & max pressure for different flow models.

Notably, as the inlet velocity increases from 50 m/s to 100 m/s and then to 200 m/s, the Maximum velocity within the car's airflow exhibits a substantial rise. This increment results from the heightened kinetic energy introduced into the system with higher inlet velocities. The faster-moving air at elevated inlet speeds carries more kinetic energy, contributing to an overall increase in airflow velocity within and around the car. This phenomenon can be visualized in Figure 5.

The choice of the RANS (k-ω SST) turbulent model is pivotal in understanding the observed changes. This model incorporates the consideration of turbulent kinetic energy (k) and the specific rate of dissipation (ω) to accurately simulate turbulent flow. At higher inlet velocities, the model responds by accounting for the greater kinetic energy input. This heightened turbulence, driven by the increased kinetic energy, substantially contributes to an augmented maximum velocity within the car's airflow.

The increase in inlet velocity introduces a surplus of kinetic energy into the system, intensifying the turbulence within the flow. This heightened turbulence, in turn, leads to a more energetic and dynamic airflow around the car. The "RANS (k-ω SST)" turbulent model adeptly captures and represents this intensified turbulence, providing a realistic portrayal of the aerodynamic conditions. The observed rise in maximum velocity can be attributed to the interplay between higher inlet velocities, the increased kinetic energy introduced into the system, and the corresponding intensification of turbulence. The graph effectively illustrates how the F1 car's aerodynamics respond to varying inlet velocities, reflecting the dynamic nature of the airflow and the energy imparted by the faster-moving inlet air.

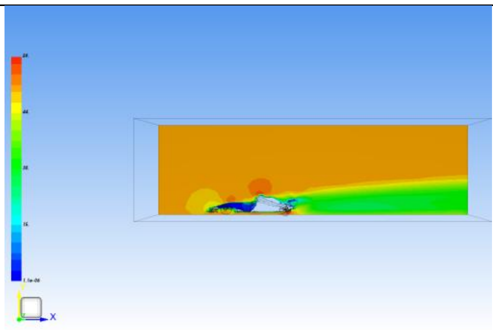
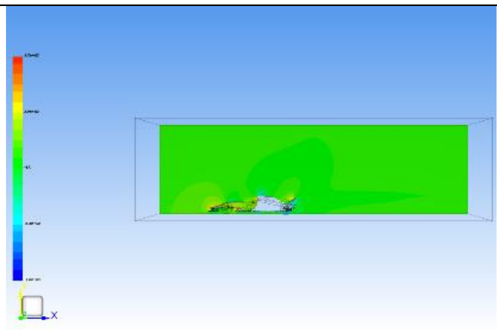
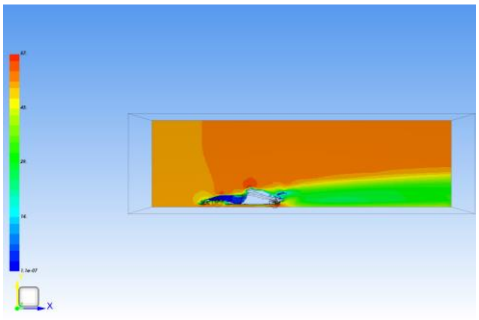
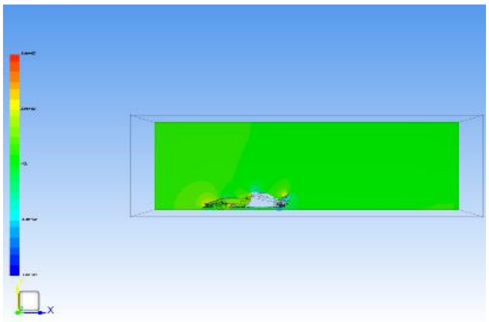
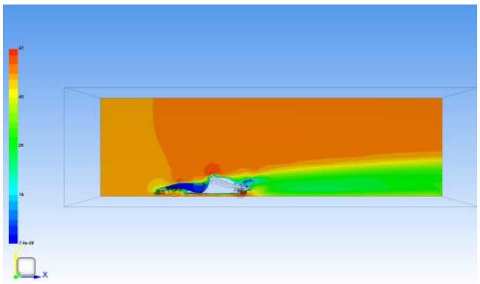
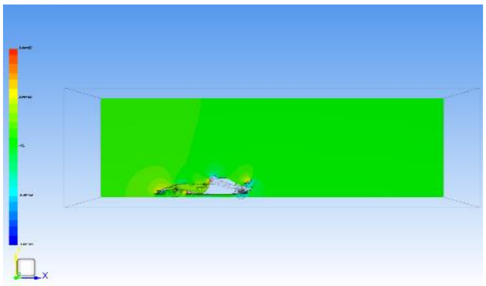
Model	Maximum Velocity (m/s)	Maximum Pressure (Pa)
Laminar		
RANS (k- ε)		
RANS (k-ω SST)		

Figure 5: Inlet velocity at 50 m/s with different flow models.

4. CONCLUSION

The CFD simulation has been successfully carried out using SimFlow software. The different inlet velocities were investigated using the RANS (k-ω SST) turbulent model. The analysis emphasized the pivotal role of inlet velocity in achieving peak car speed performance. By systematically varying the boundary conditions, we identified specific areas of the car that contribute to the highest maximum pressure, offering a nuanced understanding of the interplay between car speed and air friction. This knowledge is instrumental in optimizing the external aerodynamics of the car body design for enhanced performance. Besides, in the second analysis, the inlet velocity was fixed at 50 m/s. We examined various flow models: Laminar, RANS (k-ε), and RANS (k-ω SST)- underscored turbulence models' significance in capturing the intricacies of external flow dynamics. The laminar model, with its assumption of smooth flow, showcased both higher velocities and pressures. Introducing turbulence through RANS models altered flow patterns, impacting maximum velocity and maximum pressure. This highlights the critical importance of selecting an appropriate turbulence model for accurate representation of external flow complexities. These cases collectively demonstrate the versatility and efficacy of CFD methods in addressing external aerodynamic aspects comprehensively. These findings are helpful for setting up boundary conditions, understanding turbulence effects, and ultimately optimizing vehicles' aerodynamic performance. The knowledge gained contributes to the ongoing advancements in F1 car design, enhancing efficiency and competitiveness on the racetrack.

ACKNOWLEDGEMENTS

The author sincerely appreciates the Faculty of Mechanical Engineering & Technology's invaluable support and resources throughout this research. Thanks to the coordinator for the CFD course for their guidance and mentorship.

REFERENCES

[1] Spalart, P. R. Strategies for turbulence modelling and simulations. International journal of heat and fluid flow, vol 21, issue 3 (2000) pp. 252-263.

[2] Toubiana, E., Russeil, S., Bougeard, D., & François, N. Large Eddy simulation and Reynolds-averaged Navier–Stokes modeling of flow in staggered plate arrays: Comparison at various flow regimes. Applied Thermal Engineering, vol 89, (2015) pp. 405-420.

[3] Cliquet, J., Houdeville, R., & Arnal, D. Application of laminar-turbulent transition criteria in Navier-Stokes computations. AIAA journal, vol 46, issue 5 (2008) pp. 1182-1190.

[4] Heinz, S. A review of hybrid RANS-LES methods for turbulent flows: Concepts and applications. Progress in Aerospace Sciences, vol 114, (2020) p. 100597.

[5] Tang, J., Cui, P., Li, B., Zhang, Y., & Si, H. Parallel hybrid mesh adaptation by refinement and coarsening. Graphical models, vol 111, (2020) p. 101084.

[6] Pawar, S., & San, O. CFD Julia: A learning module structuring an introductory course on computational fluid dynamics. Fluids, vol 4, issue 3 (2019) p. 159.

[7] Gayer, M., Kortelainen, J., & Karhela, T. CFD modelling as an integrated part of multi-level simulation of process plants: semantic modelling approach. In Proceedings of the 2010 summer computer simulation conference, (2010) pp. 219-227.

[8] Tu, J., Yeoh, G. H., Liu, C., & Tao, Y. Computational fluid dynamics: a practical approach. Elsevier. (2023)

[9] Elzamar, M. A. M. M. Computational Fluid Dynamics Analysis of Tesla Cybertruck.

[10] Zhang, Z., Wang, Q., Song, S., Zhang, C., Ren, L., & Zhang, Y. Joint research on aerodynamic characteristics and handling stability of racing car under different body attitudes. Energies, vol 15, issue 1 (2022) p. 393.

[11] Sharp, R. S., & Peng, H. Vehicle dynamics applications of optimal control theory. Vehicle System Dynamics, vol 49, issue 7 (2011) pp. 1073-1111.

[12] Kamal, M. N. F., Ishak, I. A., Darlis, N., Maji, D. S. B., Sukiman, S. L., Abd Rashid, R., & Azizul, M. A. A review of aerodynamics influence on various car model geometry through CFD techniques. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, vol 88, issue 1 (2021) pp. 109-125.

[13] Shah, K., Prabhakar, T. V., Sarweshkumar, C. R., & Abhishek, S. V. Construction of a digital twin framework using free and open-source software programs. IEEE Internet Computing, vol 26, issue 5 (2021) pp. 50-59.

[14] Mishra, S., Misra, A., Rao, P. R., & Rao, D. N. Simulation of Aerodynamic Flow Parameters over a Simplified Sedan Car. International Journal of Engineering, Science and Mathematics, vol 6, issue 8 (2017) pp. 1265-1274.

[15] Fu, C., Uddin, M., Robinson, C., Guzman, A., & Bailey, D. Turbulence models and model closure coefficients sensitivity of NASCAR Racecar RANS CFD aerodynamic predictions. SAE International Journal of Passenger Cars-Mechanical Systems, vol 10, (2017) pp. 330-344.

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

Author contributions statement: Conceptualization; Methodology; Software; Formal Analysis; Investigation; Writing & Editing, Y.P. Tang.