

Computational Analysis of Aerodynamic Drafting Effects in Lemans Racing Cars

Jiaqi He *

International Education College, Wuhan University of Technology, Wuhan, 430070, China

* Corresponding Author Email: moninle@whut.edu.cn

Abstract. Le Mans racing is a comprehensive motorsport event that tests not only the long-distance stability of race cars but also considers fuel economy and speed, where aerodynamics plays a crucial role. Many manufacturers place great emphasis on the aerodynamic design of race cars. However, in-depth research on the application of drafting effects in Le Mans racing and studies on team formations are somewhat lacking. Through Ansys simulations, this study deeply explores the impact of drafting on race cars and, by varying the distances between cars, identifies differences in air resistance encountered by different formations. The findings reveal that drafting can substantially decrease the aerodynamic drag experienced by race cars when they are positioned at relatively close proximities to each other. As this distance increases, a "drag bubble" phenomenon occurs within a certain range, where the trailing cars begin to experience increased resistance. On the other hand, the study of car formations under race-regulated conditions has identified optimal vehicle spacing that minimizes the average aerodynamic drag faced by the team, providing strategic insights into the arrangement of team formations for improved race performance.

Keywords: Aerodynamics, Drafting Effects, Computational Fluid Dynamics, Flow Field Interaction, Racing Car Design.

1. Introduction

In today's motorsport, a significant amount of aerodynamic technology is utilized to increase the speed of race cars. In Le Mans racing, drivers aim to cover as much distance as possible within 24 hours. By closely following team formations and utilizing drafting techniques, they can significantly reduce overall air resistance and improve fuel efficiency, allowing race cars to travel further on a single tank of fuel. Since overtaking and disrupting other cars is not overly emphasized in this event, such designs cause the following car to be in turbulent wake, thereby affecting its air resistance. In the studies by Zabat [1], Hong, [2], and Brzustowicz, [3] it was shown that the trailing car experiences an increase in resistance at certain points and endures more drag than the leading car. This characteristic has been exploited in some racing competitions to extend the advantage of the leading car, such as in F1. On the track, this phenomenon is observed as the rear vehicle starts to slow down, whereas the front vehicle pulls away, especially within this specific distance range. Jacuzzi and Granlund [4] have undertaken research aimed at minimizing these drag peaks.

Although Le Mans race cars can reach very high speeds with the throttle fully open, to improve endurance over the 24 hours and reduce the likelihood of car failures, teams generally aim for average speeds of 40-50m/s. Due to engine power restrictions specified by the FIA for certain classes, this ensures a more level playing field among different manufacturers, forcing car engineers and drivers to rely more on aerodynamic techniques, especially during long straight sprints on the track where aerodynamic drag constitutes 90% of the total resistance.[1]

In wind tunnel experiments on 1/8 scale mini-van models and 1/20 scale commercial truck models by Zabat [1], some researchers observed reductions in team drag. Significantly, some research identified a pronounced link between a vehicle's position within a formation and its aerodynamic drag or fuel efficiency [1, 5]. Vehicles positioned within the middle of a formation were discovered to encounter the lowest levels of drag and thus, realized the most substantial fuel efficiency gains. This advantage was observed to be most pronounced at separation distances ranging from 3 to 6 meters, diminishing outside of this range.

In other competitions, drafting techniques are also used to enhance the level of competition, such as in cycling races, marathons [6], etc. The collaborative research and analysis conducted by various scholars [7-10] have significantly deepened the understanding of aerodynamic interactions during drafting, particularly in cycling. Utilizing a diverse array of investigative approaches, including field experiments, wind tunnel tests, and Computational Fluid Dynamics (CFD) simulations, these studies have collectively illuminated the drafting phenomenon. A key insight from this body of work is the observation that aerodynamic drag on cyclists following behind others decreases as the distance to the leading cyclist reduces. Specifically, through CFD simulations, discovered that the leading cyclist mitigates the high-pressure area in front of the trailing cyclist and the low-pressure zone behind them, effectively lessening the aerodynamic drag faced by the cyclist in the rear. Moreover, the trailing cyclist by filling up the vacuum behind the leading cyclist, thereby reduces the drag for the leading cyclist [8]. While traditional studies indicate that the distance between vehicles can affect drag reduction, these studies often do not provide detailed quantitative data. There is a lack of specific explanation for the relationship between the exact distance between vehicles and the effect of drag reduction, as well as the underlying physical mechanisms, leaving a gap in the deep understanding of the specific mechanisms of drag reduction effects.

Considering that the Le Mans race typically involves teams of two to three cars competing together, this study specifically focuses on the aerodynamic characteristics of race cars under slipstreaming conditions, configured as teams of two and three cars. By employing CFD simulations to construct models of two different team formations, this research delves into how the relative positions of vehicles affect their aerodynamic performance, as well as the potential impact of this interaction on vehicle speed and fuel efficiency.

The significant value of this study lies in its innovation in the method of studying race car aerodynamics. Utilizing CFD simulation technology allows for the exploration of race car aerodynamic behavior in actual racing environments without the limitations of physical testing, providing a more accurate theoretical basis for race car design. Moreover, by analyzing the impact of different team formations on aerodynamic performance, this study offers scientific evidence for racing teams to develop more effective racing strategies.

This approach not only enhances the understanding of aerodynamics involved in high-speed racing but also paves the way for advancements in targeted race car design and racing strategies. The findings of this study could lead to the development of race cars with higher aerodynamic efficiency, capable of maintaining high speeds while reducing fuel consumption, ultimately providing competitive advantages for racing teams in the Le Mans event.

2. Methodology

2.1. Simulation Parameters

To delve into the intricacies of race car aerodynamics, this study focuses on the 24 Hours of Le Mans race held in June 2023 at the renowned Le Mans circuit in France. This event was chosen as the primary subject for its wealth of data on race car aerodynamics. In this research, a special race car model was designed to run on the Le Mans track at a constant speed of 40 m/s. This speed setting is based on the average speed of cars in actual races. To concentrate on the aerodynamic effects, the track conditions were simplified to an ideal dry state, and the ambient temperature was set at 24°C. This temperature setting considers the impact of temperature and humidity on air density, thereby influencing the aerodynamic characteristics of race cars [11]. Specific parameters are shown in Table 1.

Table 1. Summary of fixed values

Name	Value
Velocity, v (m/s)	40
Length, L_c (m)	3500
Density of Air, ρ (kg/m ³)	1.204
Dynamic Viscosity of Air, μ (Ns/m ²)	1.81×10^{-5}
Reynolds Number, Re	9312707

2.2. Car Modelling

In this study, a classic enclosed-wheel race car equipped with a rear wing is utilized, aligning with the typical design characteristics of the Le Mans Prototype (LMP) class vehicles used in the Le Mans race. To reduce computational time for car simulations, the race car model was partially simplified while retaining its key aerodynamic features shown in Figure 1. This balance ensures that the model provides accurate aerodynamic data while optimizing the use of computational resources. Non-critical components like the engine system, cooling system, and the intricate details of the transmission and suspension systems were simplified or omitted in some cases, as they do not affect the external aerodynamics of the car. Important race car model parameters are shown in Table 2.

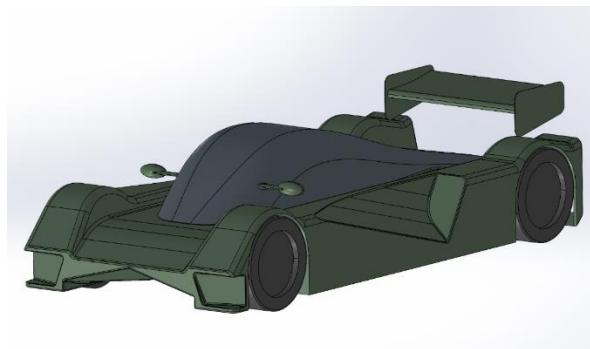


Figure 1. LMP Model

Table 2. External dimensions of the car

Name	Vale
Car's Length (mm)	3500
Car's Width (mm)	1600
Spoiler's Length (mm)	975
Spoiler's Width (mm)	450

The rear wing plays a significant role in aerodynamics, offering increased downforce by altering the airflow around the car's rear. Most manufacturers in the LMP category choose to use the largest rear wing allowed under FIA regulations to balance the effect of reduced car weight.

The simulation involved setting various distances between vehicles to analyze the airflow distribution. Through the comparison of simulation data across various vehicle spacings, the influence of the leading car's drag on the car behind was evaluated. Comparisons between simulations involving two and three cars were conducted to gauge the effects of varying formations on the trailing vehicle. Emphasis was placed on examining the airflow characteristics in the wake of the vehicles and the aerodynamic impacts at diverse separations.

The computational domain was carefully sized to ensure that the boundaries did not interfere with the flow around the vehicles, maintaining the accuracy of the simulations. This approach allowed for the application of outlet boundary conditions without disturbing the airflow characteristics in the vehicle's wake. Finer mesh grids were used around the vehicle and in other key areas, such as the wake of the rear wing and diffuser, to capture complex flow phenomena and vortex formation. By

setting appropriate turbulence parameters, the simulation reflected the real-world operating environment of high-speed racing cars, providing realistic test conditions for design. The meticulously designed grid and turbulence model enabled the study to accurately capture boundary layer phenomena and the characteristics of the vehicle wake, which is crucial for understanding and optimizing the aerodynamic performance of race cars.

2.3. Car Meshing

This research utilized ANSYS FLUENT 2022 R1 for CFD simulations, a widely used advanced tool for fluid mechanics studies. The simulation environment was set up as a free-stream test environment, replicating the aerodynamic behavior of real race cars. The blockage ratio analysis for individual car models showed a 5.6% blockage ratio, which is similar to the standard 5% blockage ratio often used in wind tunnel tests [12-14]. Referencing the study by Choi and Kwon [15], even an increase in blockage ratio to 10% has minimal impact on the accuracy of test results. Therefore, this study could safely disregard any errors caused by the blockage ratio.

In the configuration of the grid and boundary conditions, this investigation established a velocity inlet at a distance of 5 meters in front of the leading vehicle, mirroring the typical spacing observed among race cars in actual competitions. Positioned 10 meters beyond the final car, as depicted in Figure 2, was the pressure outlet. The central concern of this study revolved around the aerodynamic interference among team cars, with a specific emphasis on how such interference could lead to diminished drag forces. To this end, the study deliberately excluded the consideration of airflow underneath the vehicles, focusing instead on the aerodynamic dynamics around and between them. The dimensions for the computational domain were carefully chosen to ensure that the domain's boundaries would not affect the flow dynamics around the car model being examined, as illustrated in Figure 2.

This research utilizes the $k-\epsilon$ turbulence model along with non-equilibrium wall functions for conducting computational analysis of steady viscous fluid flow. This model has been validated as suitable for the analysis of flow fields around vehicles [16]. Given that the flow's Mach number is below 0.3, indicating the flow is incompressible, the use of Reynolds-Averaged Navier-Stokes (RANS) equations to control the incompressible turbulent flow is thus a reasonable assumption.

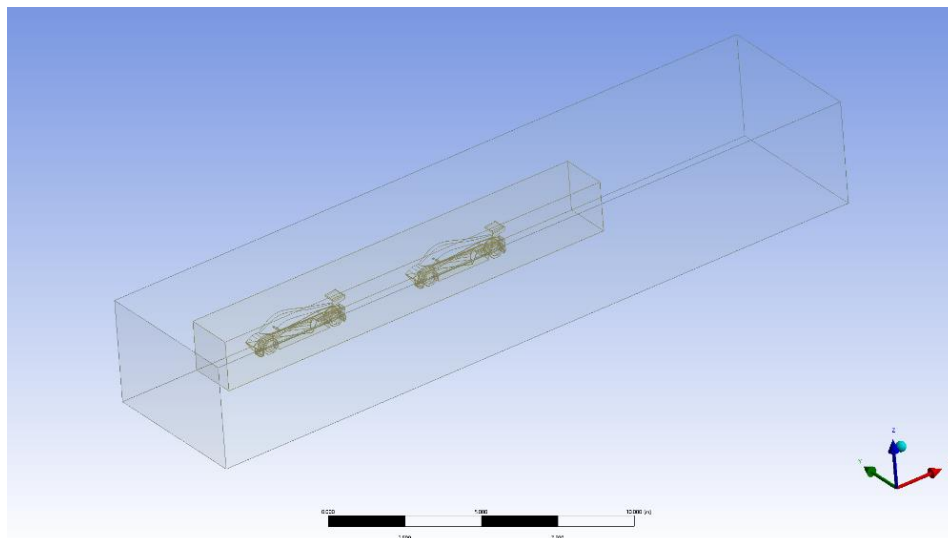


Figure 2. Example of constructing a separate racing grid

In Figure 3, the grid division of key parts of the race car can be seen. The fine and reasonable grid division plays a very important role in subsequent simulations.



Figure 3. Mesh model of the isolated car

In this study, a standardized set of model parameters was established to control the variables in subsequent simulations. These parameters for the race car were fixed as shown in Table 3.

Table 3. Standardized set of the model

Domain	Nodes	Elements
fluid	20088264	16185511

3. Results

3.1. Isolated Car

To compare with the data of the following vehicles, this study first conducted a series of independent race car simulation tests. The results of these tests are illustrated in the accompanying Figure 4. For consistency in subsequent data, all simulation parameters in this study were aligned with those used in the independent car simulation tests.

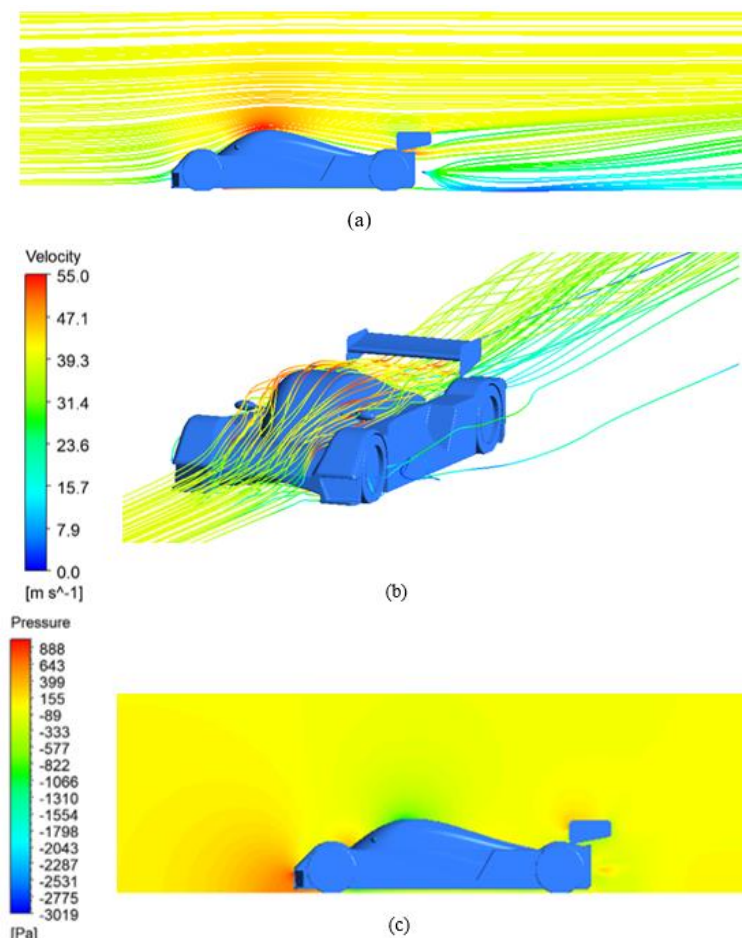


Figure 4. Isolated car: (a). Streamlines on the XZ plane, (b) 3D streamlines of the velocity, and (c) Contour pressure plots on the XZ plane

The streamlined diagrams illustrate the path of the airflow over the surface of the race car. Comparing these with the air velocity maps a noticeable reduction in airflow speed can be observed in certain areas at the rear of the car.

Figure 4 illustrates that air velocity increases as it passes over the car's top, then experiences a pronounced drop in the vortex zone at the vehicle's rear. Additionally, the figure reveals a "tunnel" of reduced air velocity directly behind the car. This area is particularly relevant in wake studies, suggesting that positioning trailing vehicles within this "tunnel" could significantly impact their aerodynamic efficiency.

Furthermore, the contour map in Figure 4 highlights areas of high pressure around key components of the car, such as the front windshield, windscreen, spoiler, and the area around the rear bumper. These high-pressure zones are critical in understanding the overall aerodynamic performance of the car, as they directly influence aspects like drag and downforce, which are vital in race car design and performance. Understanding how these high-pressure areas interact with the airflow and the vehicle's wake can provide valuable insights for optimizing the car's aerodynamic efficiency.

Based on the simulation with Ansys, the parameters in Table 4 can be obtained as reference parameters for subsequent studies.

Table 4. Standardized set of the model

Drag Force	Drag Ratio
481.7 N	0.3779

3.2. Wake Effects at Different Vehicle Spacings

In the two-car team mode simulation conducted using ANSYS, the following streamlined diagrams were obtained. In Figures 5 and 6, the process of the leading car blocking the air is shown, with the airflow direction along the centerline plane. In this context, x represents the distance between vehicles, and L denotes the length of a single vehicle. At $x/L=0.25$, a vortex appears at the front of the trailing car, which is visible in Figure 6. The space between the rear of the leading car and the front of the trailing car is largely occupied by this vortex, which markedly influences the air resistance encountered by the trailing car. Figure 7 illustrates how variations in vehicle spacing affect both drag and the drag ratio. Due to the simulation data requiring long computational times to obtain, this study employs an exponential function Formula (1) to fit the data:

$$y = a + b \times e^{-\frac{c}{x}} \quad (1)$$

The parameter values were determined by employing least squares optimization, utilizing the Levenberg–Marquardt algorithm for this estimation process. The algorithm was initially proposed by Kenneth Levenberg [17] in 1944 and later refined by Donald Marquardt [18] in 1963.

This study observed an increase in drag on the trailing car under the influence of the "drag bubble," but due to differences in the structural design of the cars, it did not find the situation as Edwin Chern Junn Gan [19] observed in NASCAR racing, where the trailing car experiences more air resistance than the leading car. This study only observed the same trend, noting an increase in drag on the trailing car. However, as the distance between the cars continuously increases, this trend of increasing drag does not persist for a long time.

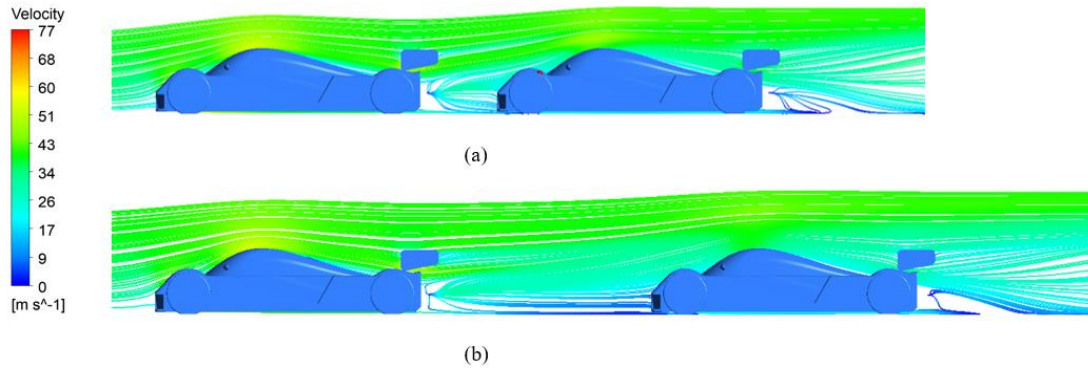


Figure 5. Streamline on the XZ plane for the 2-car platoon: (a) $x/L=0.25$, (b) $x/L=0.8$

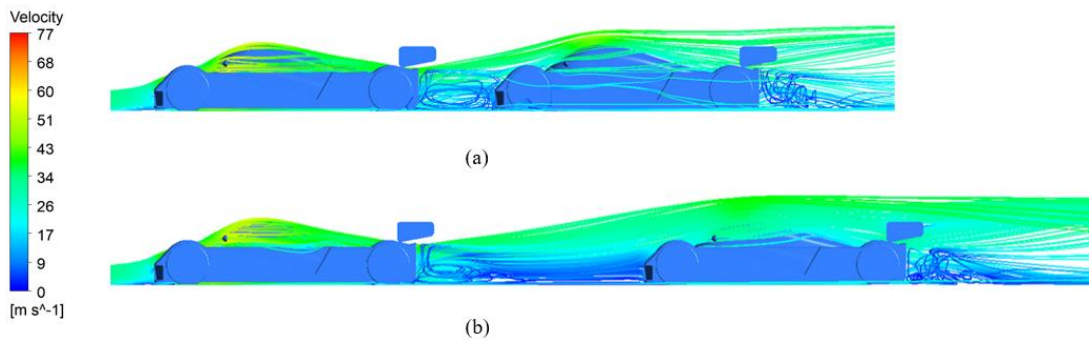


Figure 6. 3D streamlines on the XZ plane for the 2-car platoon: (a) $x/L=0.25$, (b) $x/L=0.8$

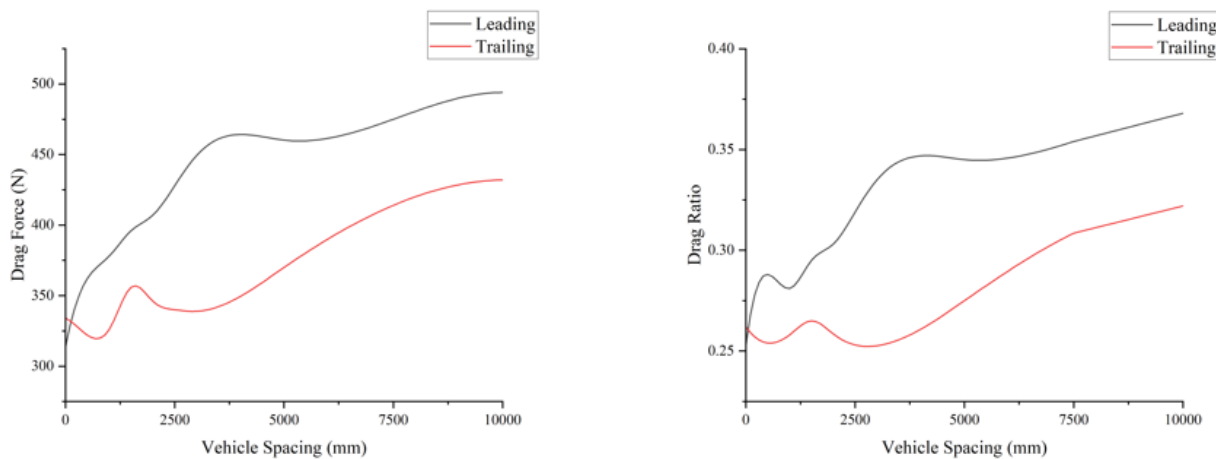


Figure 7. The Drag Force and the Drag ratio for the 2-car platoon

3.3. Drag Reduction in Various 3-Car Platoon

This study investigates the impact of race car formations on the potential for drag reduction, with a particular focus on how changing the distances between members within the formation affects the air resistance experienced by each team member.

In the analysis of linear car formations, it was observed that the wake vortices generated by the middle vehicles create a strong circulation of airflow, which significantly affects the aerodynamic characteristics of the third car, leading to a substantial increase in aerodynamic drag. This increase is notably greater than that experienced by the leading car, and in some cases, exceeds the drag modeled in single-vehicle simulations. The diagrams distinctly illustrate the variation in the wake produced by the leading race car compared to that of the second race car when the gap between the front and rear vehicles is 2 meters. From Figures 8 to 10, the wake generated by the second car is more irregular and chaotic compared to that generated by the first car, causing the following car to be severely

affected by this part of the airflow, thereby increasing aerodynamic drag. At the same time, this also correspondingly affects the fuel economy of the race car.

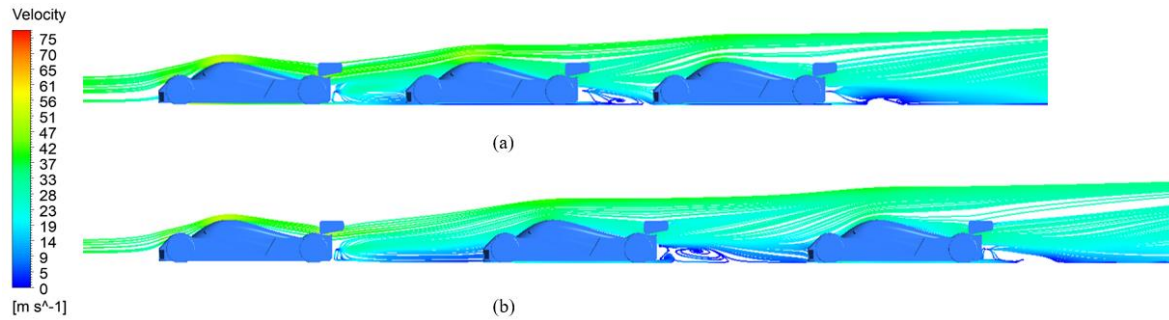


Figure 8. Streamline on the XZ plane for a 3-car platoon in linear formation: (a) $x/L=0.25$, (b) $x/L=0.5$

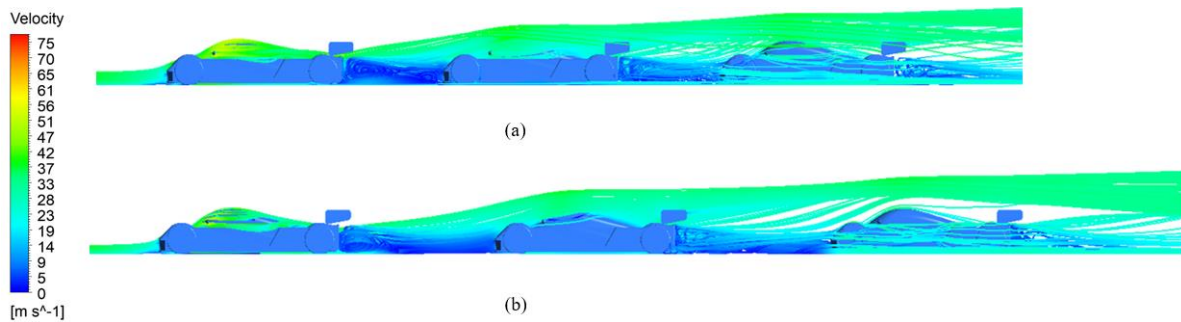


Figure 9. 3D streamlines on the XZ plane for a 3-car platoon in linear formation: (a) $x/L=0.4$, (b) $x/L=0.5$

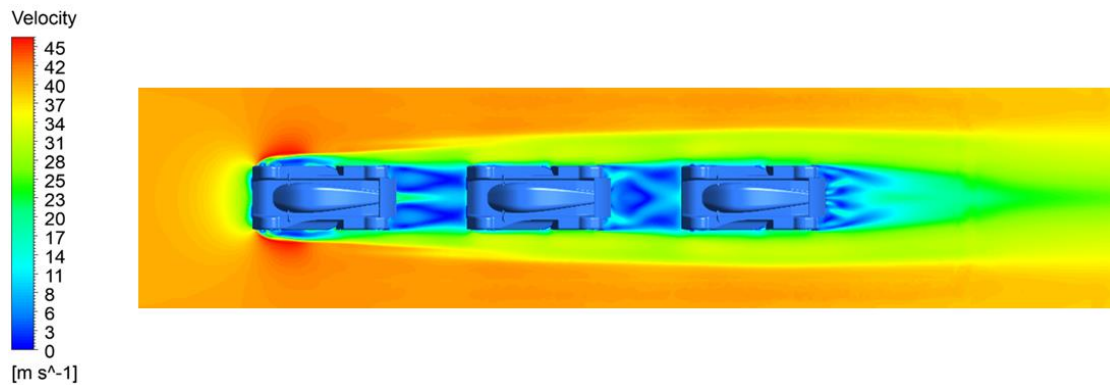


Figure 10. Streamwise velocity plot on the XY plane ($Z=0.2\text{m}$) for a 3-car platoon in a linear formation

When analyzing the inverted V formation, as shown in Figure 11, the presence of two front cars does not create a significant "drag bubble" effect due to the gap between the front cars allowing airflow to pass through. This leads to a significant decrease in aerodynamic drag for the trailing car, in contrast to simulations involving a car in isolation. Given that the leading vehicles do not obstruct all the airflow, there's a substantial region of high stagnation pressure directly in front of the trailing car, as highlighted by the blue-colored airflow cluster in Figure 12. Additionally, the leakage of airflow between the front cars leads to significant pressure peaks at the center of the trailing car, which also affects the aerodynamic performance of the entire team.

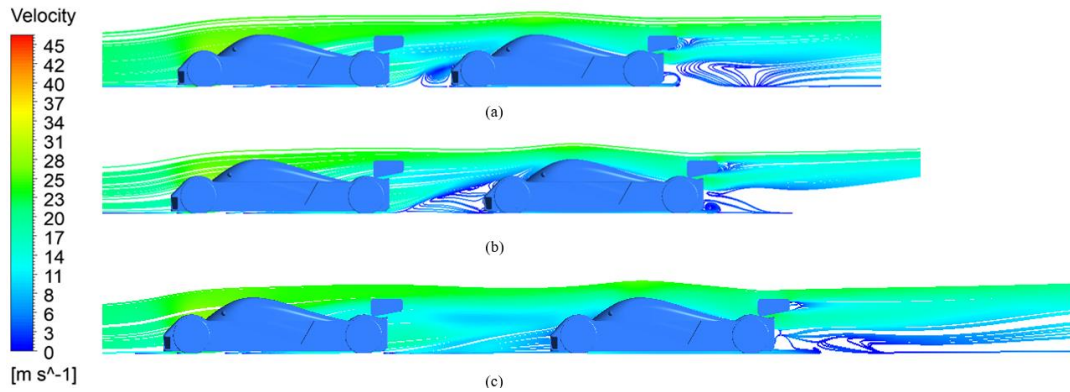


Figure 11. Streamline on the XZ plane for the 3-car platoon in a V formation: (a) $x/L=0.25$, (b) $x/L=0.5$, (c) $x/L=0.8$

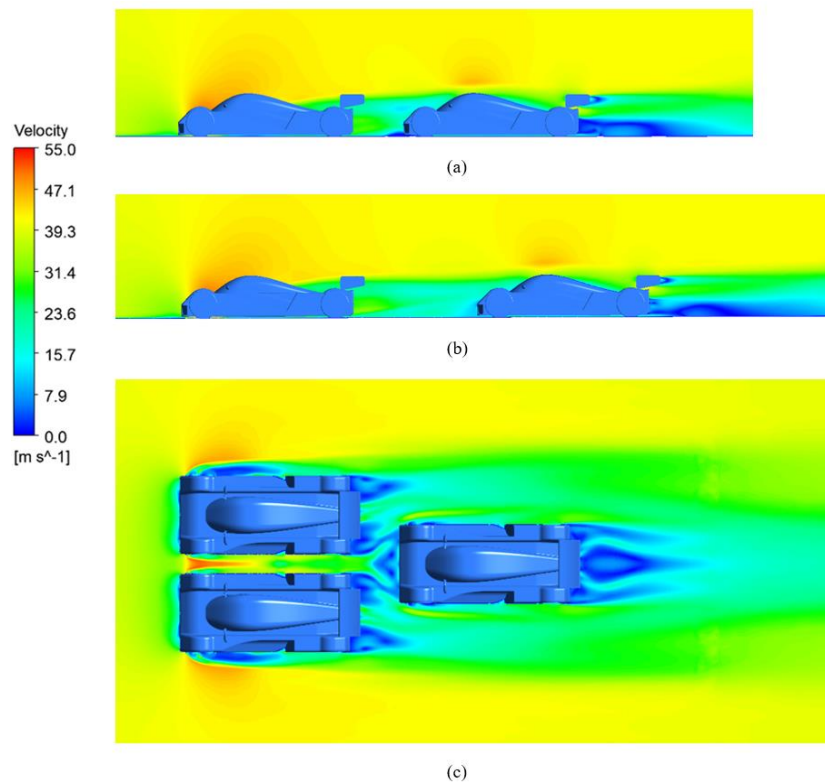


Figure 12. Streamwise velocity plot for the 3-car platoon in a V formation: (a) On the XZ plane - $x/L=0.25$, (b) On the XZ plane - $x/L=0.8$, and (c) On XY plane ($Z=0.2m$) - $x/L=0.25$

Figure 13 indicates that slipstreaming offers the most advantage to the car following behind at reduced distances between vehicles, where it encounters the minimum amount of drag within this specified range. Within the 1000-2000 range, all members of the team begin to experience a gradual increase in drag, but the drag on the last car changes very little. This phenomenon, detailed in the studies by Jacuzzi and Granlund [4], arises from the "drag bubble" generated by the front car, leading to an increase in drag for the trailing car due to strong flow recirculation within the bubble. The leading car has a noticeable aerodynamic advantage over the trailing car in this scenario, allowing it to pull away from the car behind more quickly. Research [1-3] has demonstrated that slight modifications to vehicle geometry can substantially impact the aerodynamic drag experienced by both leading and trailing cars.

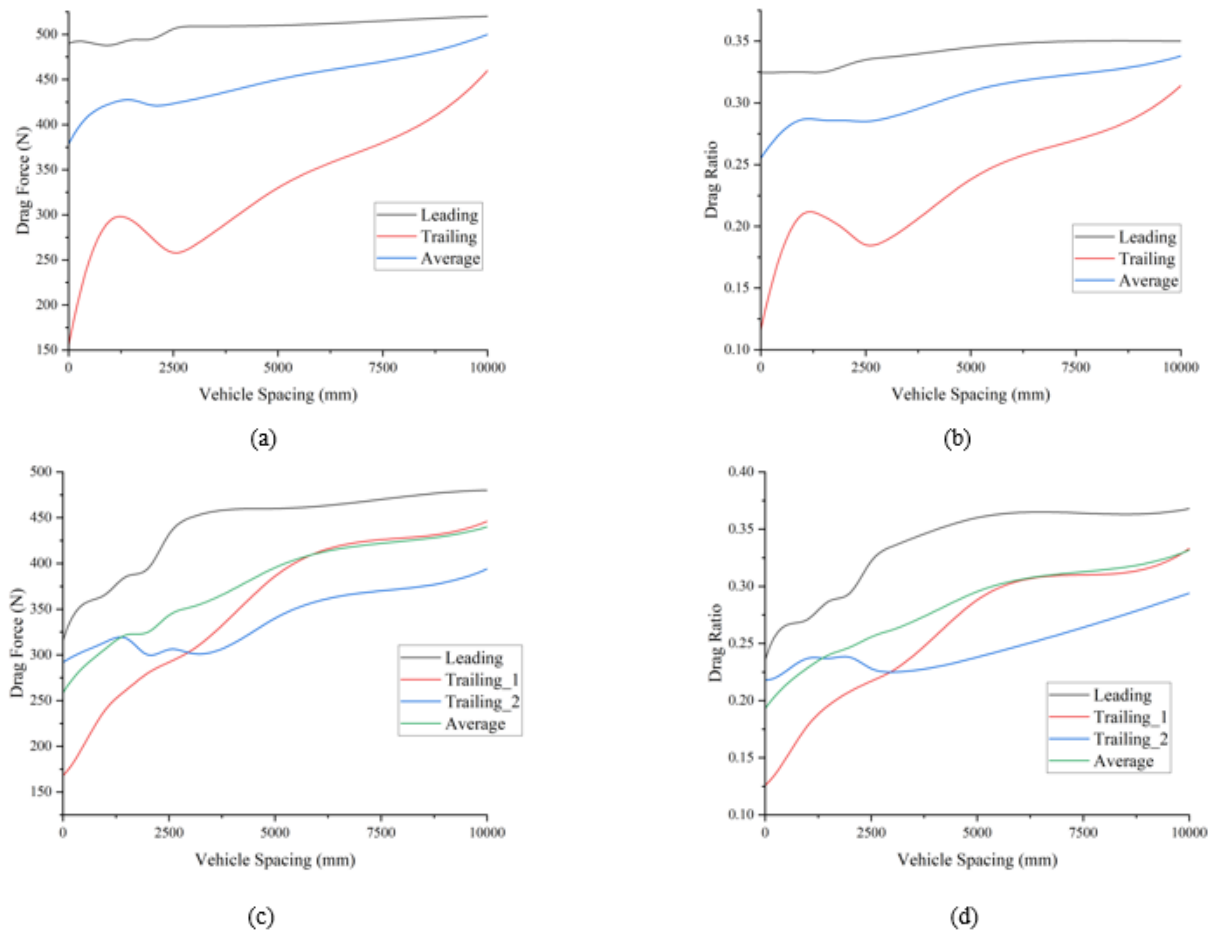


Figure 13. 3-car platoon in V and linear formation: (a) Drag Force of the V Formation (b) Drag Ratio of V Formation (c) Drag Force of the Linear Formation (d) Drag Ratio of the Linear Formation

Nonetheless, the graph illustrates that as the gap between the leading and trailing cars widens, there is a noticeable diminishing trend in the drag experienced by the trailing car, particularly within the 2000-4000 mm range. This reduction occurs as the trailing car gradually exits the area affected by the "drag bubble" generated by the leading car, no longer influenced by this area, leading to a gradual decrease in aerodynamic drag. Within the $x > 3000$ vehicle spacing range, it is observed that since the leading car still directs a portion of the airflow upward to some extent, the aerodynamic drag experienced by the trailing car is slightly less than that of the leading car, eventually forming a situation where the aerodynamic drag of the 1st car $>$ the 2nd car $>$ the 3rd car. As the distance further increases, the three cars gradually revert to experiencing air resistance like that of a car studied independently. This indicates that when $x/L > 3$, the leading car no longer affects the trailing car as if the two cars are experiencing air resistance as they would when driving independently on the track. The air resistance experienced by the cars almost shows a linear change after reaching this area.

4. Conclusion

In this research, Ansys was used to simulate and calculate models with different vehicle spacings and racing formations, and an in-depth study of the slipstreaming effect was conducted. Based on the slipstreaming effects under different platoon configurations, corresponding optimal racing strategies were obtained, which can help teams more advantageously analyze the fuel economy and overtaking capabilities of their race cars.

Through the examination of aerodynamic interactions between two race cars influenced by slipstreaming at varying vehicle distances, the subsequent conclusions were reached:

1). The slipstreaming effect proves to be most advantageous for the trailing car when the separation between the two vehicles is less than or equal to a ratio of $x/L=0.25$.

2). When the vehicle spacing is within the range of $x/L=0.5-2$, the air resistance experienced by the trailing car significantly increases.

3). In the vehicle spacing stage of $x/L=0.25-1$, due to the impact of the "drag bubble," the trailing car undergoes a process of air resistance first increasing and then decreasing. This is because as the vehicle spacing gradually increases, the trailing car gradually moves out of the area affected by the "drag bubble" generated by the leading car.

By studying the aerodynamic interaction between three race cars under the slipstreaming effect at different vehicle spacings, the following conclusions can be drawn:

1). According to different racing situations, if the team needs to help only one race car to finish the race faster, a V-shaped formation should be used, and the distance of the trailing car should be controlled at around $x/L=0.7$. At this distance, the air resistance experienced by the race car is lower, while also ensuring a safe reaction distance in case of sudden incidents with the leading car.

2). When the team needs all the race cars to maintain a low air resistance to increase endurance, a linear formation should be chosen, and the spacing between the cars should be controlled at around 2000mm or $x/L=0.55-x/L=1$. At this spacing, the overall team's average air resistance is lower, which is more beneficial for the whole team to maintain high speed while also reducing fuel consumption rates.

While this study provides an in-depth investigation into the effects of vehicle spacing and race car formations on slipstreaming, offering significant theoretical support for racing strategies, its reliance on Ansys software for simulation analysis inevitably comes with certain limitations. Specifically, the simulation environment may not fully replicate the complex conditions encountered in real races, such as weather changes, actual wear and tear on race cars, and the immediate reactions and decisions of drivers. Therefore, although simulation results provide a theoretical foundation, the practical application requires verification and strategy adjustments through on-site real-car testing to ensure their effectiveness in a real racing environment.

Future research directions should focus on comparing simulation results with real car testing to validate the accuracy and practical value of the simulation models. Through actual testing, the specific effects of slipstreaming under different environmental conditions can be more accurately assessed, considering multiple factors such as track type, climate changes, and vehicle performance. Further exploration of more race car formations and vehicle spacing configurations aims to enhance overall team performance and fuel efficiency. These studies not only provide scientific and practical strategy recommendations for racing teams but also help gain a competitive edge in highly competitive events.

References

- [1] Zabat Michael, Nick Stabile, Stefano Frascaroli, and Fred Browand. Drag forces experienced by 2, 3 and 4-vehicle platoons at close spacings. SAE transactions, 1173 - 1181 (1995).
- [2] Hong Patrick, Bogdan Marcu, Fred Browand, and Aaron Tucker. Drag forces experienced by two, full-scale vehicles CFD Letters, 12 (7), 1 - 16 (2020).
- [3] Brzustowicz, John P., Todd H. Lounsberry, and Jean-Michel Esclafer de La Rode. Experimental & Computational simulations utilized during the aerodynamic development of the dodge intrepid R/T race car. SAE Transactions, 2387 - 2403 (2002).
- [4] Jacuzzi, Eric, and Kenneth Granlund. Passive flow control for drag reduction in vehicle platoons. Journal of Wind Engineering and Industrial Aerodynamics, 189, 104 - 117 (2019).
- [5] Michaelian, Mark, and Fred Browand. Quantifying platoon fuel savings: 1999 field experiments. SAE transactions, 1401 - 1410, (2001).
- [6] Lukas Schickhofer, Henry Hanson. Aerodynamic effects and performance improvements of running in drafting formations. Journal of Biomechanics, 122, 110457, (2021).
- [7] Barry, Nathan, John Sheridan, David Burton, and Nicholas AT Brown. The effect of spatial position on the aerodynamic interactions between cyclists. Procedia Engineering 72, 774 - 779, (2014).

- [8] Blocken, Bert, Thijs Defraeye, Erwin Koninckx, Jan Carmeliet, and Peter Hessel. CFD simulations of the aerodynamic drag of two drafting cyclists. *Computers & Fluids*, 71, 435 - 445, (2013).
- [9] Edwards, Andy G., and William C. Byrnes. Aerodynamic characteristics as determinants of the drafting effect in cycling. *Medicine and science in sports and exercise*, 39 (1), 170 - 176, (2007).
- [10] Kyle, Chester R. Reduction of wind resistance and power output of racing cyclists and runners travelling in groups. *Ergonomics*, 22 (4), 387 - 397, (1979).
- [11] Munson, Bruce R., Theodore H. Okiishi, Wade W. Huebsch, and A. P. Rothmayer. *Fluid Mechanics*, SI Version. Wiley, (2013).
- [12] Sylvain Mouton, Sylvain Masseboeuf and Raffaello Mariani. Simulation of Atmospheric Boundary Layer Flow for Air Inlet Testing in F1 Wind-Tunnel, (2017).
- [13] Sylvain Mouton. Numerical Simulation of the Flow in the ONERA F1 Wind Tunnel, (2022).
- [14] Marco Boccione, Federico Cheli, Roberto Corradi, Sara Muggiasca, Gisella Tomasini. Crosswind action on rail vehicles: Wind tunnel experimental analyses, 96 (5), 584 - 610, (2008).
- [15] Choi, Chang-Koon, and Dae-Kun Kwon. Wind tunnel blockage effects on aerodynamic behavior of bluff body. *WIND STRUCT INT*, 1 (4), 351 - 364, (1998).
- [16] D. TienPhuc, G. ZhengQi and C. Zhen. Numerical simulation of the flow field around generic formula one. *Applied Fluid Mech*, 9 (1), 443 - 450, (2016).
- [17] Levenberg, K. A method for the solution of certain non-linear problems in least squares. *Quarterly of Applied Mathematics*, 2, 164 - 168, (1944).
- [18] Marquardt, D. An Algorithm for Least-Squares Estimation of Nonlinear Parameters. *Journal of the Society for Industrial and Applied Mathematics*, 11 (2), 431 - 441, (1963).
- [19] Edwin Chern Junn Gan, Mikhail Fong & Yee Luon Ng. CFD Analysis of Slipstreaming and Side Drafting Techniques Concerning Aerodynamic Drag in NASCAR Racing. *CFD Letters*, 12 (7), 1 – 16, (2021).