3rd International Conference on Electronics, Engineering, Computer Science and Applied Development (EESD 2026)

Article

Aerodynamic Optimization and Downforce Enhancement of a Formula One Rear Wing Based on Computational Fluid Dynamics

Zicheng Xiu [1,*]

[1] Xinhangdao Jinqiu International School, Qingdao, China
* Correspondence: Zicheng Xiu, Xinhangdao Jinqiu International School, Qingdao, China

Abstract: This study investigates the aerodynamic performance of a Formula One rear wing using computational fluid dynamics (CFD), with the primary aim of enhancing downforce generation while strictly maintaining overall aerodynamic efficiency. In high-performance motorsports, the rear wing plays a critical role in determining traction and cornering speeds, making its optimization essential for competitive advantage. A comprehensive three-dimensional rear wing model, encompassing the main plane, flap, and endplates, was meticulously developed and analyzed under steady incompressible turbulent flow conditions using advanced turbulence modeling. Key design parameters—such as the angle of attack, airfoil camber, flap gap dimensions, and endplate geometry—were systematically varied and optimized to identify the most effective aerodynamic configuration. The computational results indicate that the optimized wing configuration achieves a highly significant increase in downforce with only a marginal and moderate rise in aerodynamic drag, ultimately leading to a substantially improved lift-to-drag ratio. Detailed flow-field analysis reveals an enhanced surface pressure distribution, successfully delayed boundary layer flow separation, and significantly reduced tip vortex intensity. These aerodynamic improvements directly contribute to better vehicle stability, enhanced tire grip, and superior high-speed cornering performance on the track. Furthermore, the study demonstrates the profound effectiveness of multi-parameter aerodynamic optimization methodologies in motorsport engineering. The findings provide a robust theoretical and practical foundation for future research initiatives, particularly those involving complex full-vehicle aerodynamic coupling, transient flow simulations, and the application of advanced machine learning-based optimization techniques.

Keywords: formula one; rear wing; aerodynamic optimization; computational fluid dynamics; downforce enhancement; flow separation

Received: 25 March 2026
Revised: 19 May 2026
Accepted: 31 May 2026
Published: 05 June 2026

1. Introduction

Aerodynamic performance is one of the key factors affecting the competitive capability of Formula One racing cars. Unlike ordinary road vehicles, an F1 car operates under extremely high-speed conditions, where airflow around the vehicle directly influences tire grip, cornering stability, braking performance, and overall lap time [1]. Among various aerodynamic devices, the rear wing plays a particularly important role because it generates negative lift, commonly referred to as downforce, which increases the normal load on the rear tires and improves traction during high-speed cornering. Previous studies on racing vehicle aerodynamics have shown that well-designed aerodynamic components can significantly improve vehicle stability and handling performance under racing conditions.

The aerodynamic system of a racing car is usually composed of several interacting components, including the front wing, rear wing, diffuser, side pods, and underbody structures. Among them, the rear wing is one of the most direct and adjustable devices for controlling rear-axle downforce. By modifying the wing profile, angle of attack, flap arrangement, and endplate geometry, the pressure distribution around the wing can be changed, thereby affecting the balance between downforce generation and aerodynamic drag [2]. Existing aerodynamic analysis of formula-style racing cars has demonstrated that rear-wing design has a clear influence on airflow separation, wake structure, and overall vehicle performance.

However, increasing downforce is usually accompanied by an increase in aerodynamic drag. This creates a major design conflict in F1 aerodynamic optimization: the rear wing must provide sufficient downforce to improve cornering performance while avoiding excessive drag that reduces straight-line speed [1]. Therefore, rear-wing optimization is not simply a process of maximizing downforce, but rather a multi-objective design problem involving downforce enhancement, drag control, and aerodynamic efficiency improvement. Research focused on race car rear wings has confirmed that geometric parameters such as wing curvature, attack angle, and multi-element configuration have significant effects on aerodynamic force generation and lift-to-drag performance.

Computational Fluid Dynamics has become an important method for studying and optimizing racing car aerodynamics. Compared with wind tunnel testing, CFD allows researchers to visualize pressure distribution, velocity fields, flow separation, and vortex structures at relatively lower cost and with greater flexibility in parameter adjustment. In rear-wing studies, CFD can be used to compare different wing configurations and evaluate their influence on downforce and drag before physical prototype testing. Previous CFD-based studies on motorsport rear wings have shown that numerical simulation is effective for analyzing the aerodynamic behavior of main planes, flaps, and endplates under high-speed flow conditions [3].

In recent years, CFD has also been increasingly applied to advanced rear-wing configurations, including active rear-wing systems and adjustable aerodynamic devices. These studies indicate that rear-wing performance can be further improved by optimizing structural parameters according to different driving conditions. For Formula One racing cars, such optimization is especially meaningful because small aerodynamic improvements can lead to measurable gains in stability, cornering speed, and lap performance. Therefore, this paper focuses on the aerodynamic optimization of an F1 rear wing based on CFD simulation. The main objective is to analyze the flow characteristics around the rear wing, compare the aerodynamic performance before and after optimization, and propose a rear-wing design that enhances downforce while maintaining reasonable aerodynamic efficiency [4].

2. Literature Review

Research on racing car aerodynamics has increasingly emphasized the optimization of rear-wing structures because the rear wing directly affects downforce, drag, and vehicle stability. Studies on the optimization of a two-dimensional multi-element rear wing for a Formula 1 car have shown that multi-element wing configuration is an effective way to enhance aerodynamic loading while maintaining a controllable drag level. This indicates that rear-wing optimization should not only focus on a single wing profile but also consider the interaction between the main element and flap, as the gap and overlap between elements strongly influence pressure distribution and flow separation [5].

Active aerodynamic devices have also become an important direction in motorsport aerodynamic research. Reviews of drag reduction systems (DRS) and other adjustable aerodynamic technologies highlight that variable rear-wing configurations can effectively balance downforce and aerodynamic drag under different driving conditions. Research emphasizes that aerodynamic performance is not static but can be dynamically regulated according to speed requirements, cornering demands, and straight-line efficiency [6].

Such systems enable race cars to reduce drag on straights while restoring downforce during cornering, thereby improving overall lap performance. However, the implementation of active aerodynamic devices also introduces additional structural complexity and is often constrained by strict racing regulations, particularly in Formula One.

The installation position of the rear wing is another important factor affecting aerodynamic performance. Computational fluid dynamics (CFD) analyses of rear-wing location on racing vehicles have found that the relative position of the rear wing can influence the wake structure and the amount of useful downforce generated at the rear axle. This suggests that rear-wing design should not be isolated from the vehicle body [1]. Instead, the aerodynamic interaction between the wing and the upstream flow from the vehicle body must be considered during optimization.

In addition to position, the selection of airfoil profile and angle of attack plays a direct role in rear-wing performance [7]. Investigations into the aerodynamic performance of rear wings using specific airfoil profiles have shown that changes in angle of attack can significantly affect lift and drag characteristics. This finding supports the view that attack angle is one of the most practical design variables in rear-wing optimization. Nevertheless, excessive attack angle may lead to flow separation, which can reduce aerodynamic efficiency and increase drag.

Drag reduction is also a major concern in rear-wing design [8]. Studies on aerodynamic drag reduction strategies for racing vehicles have discussed the relationship between wing adjustment, drag reduction, and overall vehicle performance. This work indicates that a high-downforce configuration is not always optimal, especially on tracks with long straights where excessive drag can reduce maximum speed. Therefore, the design of a rear wing should balance downforce enhancement with drag control.

CFD has become a standard tool for analyzing racing wing aerodynamics because it allows detailed observation of flow structures that are difficult to capture through simple theoretical calculations. Research has analyzed the behavior of wings under different aerodynamic conditions, including free-stream flow and ground-effect-related environments [9]. This type of research confirms that CFD can provide useful information about pressure distribution, velocity changes, separation regions, and wake development around racing wings.

Recent research has also begun to combine CFD with intelligent optimization methods. AI-assisted CFD optimization methods for the multi-element wing angle of racing vehicles have demonstrated that artificial intelligence can improve the efficiency of aerodynamic parameter search. This direction is important because traditional CFD-based optimization often requires many simulation cases, which can be time-consuming. AI-assisted optimization can reduce the number of repeated simulations while still identifying effective aerodynamic configurations [10].

Rear-wing design procedures have also been studied from an engineering application perspective. Analyses of the design and aerodynamic performance of rear wings have provided complete workflows involving geometric design, CFD simulation, and performance evaluation. Such studies are useful for this paper because they show how a rear-wing model can be developed from initial geometry to numerical evaluation [1]. However, many studies on student-level racing vehicles focus on simplified vehicle conditions, while Formula 1 rear-wing optimization requires more attention to high-speed flow and stricter aerodynamic efficiency.

Multi-fidelity optimization is another promising method for improving rear-wing design. Multi-fidelity optimization procedures for two-element Formula 1 rear wings combine different levels of simulation accuracy to improve optimization efficiency [11]. This method is relevant because high-resolution CFD simulations are computationally expensive, especially when many design variables are considered. Multi-fidelity strategies can reduce computational cost while preserving the reliability of the optimization process.

More recent studies have continued to examine the influence of rear-wing angle on aerodynamic performance [2]. Research on rear-wing angle and vehicle aerodynamic performance has shown that adjusting the rear-wing angle can directly change drag coefficient and lift coefficient. This confirms that angle of attack remains one of the most important and practical parameters in rear-wing optimization. Overall, existing literature shows that rear-wing aerodynamic performance is affected by multiple coupled factors, including wing profile, multi-element configuration, attack angle, installation position, and optimization method. However, many studies focus on isolated parameters or simplified racing vehicles. Therefore, further research is needed to conduct a systematic CFD-based optimization of an F1 rear wing with the objective of improving downforce while maintaining acceptable aerodynamic efficiency.

3. CFD Modeling and Aerodynamic Analysis

3.1. Geometric Model and Computational Domain

The geometric model of the Formula One rear wing was established as the basic object for CFD simulation and aerodynamic analysis. The rear wing model primarily consisted of a main plane, an upper flap, two side endplates, and simplified supporting structures [12]. The main plane was designed as the primary load-generating element, while the flap was positioned above and behind the main plane to increase flow turning and enhance the pressure difference between the upper and lower surfaces. The endplates were placed on both sides of the wing to reduce spanwise flow leakage and control vortex generation at the wing tips. To reduce unnecessary computational complexity, small structural details with limited influence on the global aerodynamic characteristics were simplified, while the main aerodynamic surfaces were retained.

A three-dimensional computational domain was created around the rear wing to simulate the external flow field. The domain was set large enough to avoid blockage effects and ensure that the boundary conditions did not interfere with the near-wing flow [13]. The inlet boundary was arranged upstream of the rear wing, and the outlet boundary was located sufficiently far downstream to allow the wake region and vortex structures to develop fully. The upper and side boundaries of the domain were defined as far-field or symmetry-type boundaries to reduce artificial wall effects. The ground boundary was treated as a moving wall with the same velocity as the incoming flow, so that the relative motion between the racing car and the track could be more realistically represented. For cases where only half of the model was calculated, a symmetry plane was used along the vehicle centerline to reduce computational cost.

The boundary conditions were defined according to the typical high-speed operating condition of a Formula One car. A uniform velocity inlet was applied at the front of the computational domain, while a pressure outlet was applied at the rear. The wing surfaces and endplates were defined as no-slip walls. The moving ground condition was used to reproduce the road effect, which is important for evaluating the aerodynamic performance of racing car components. These boundary settings allowed the simulation to capture the pressure distribution, flow separation, wake development, and vortex behavior around the rear wing [14].

The computational mesh was generated using an unstructured grid combined with local refinement near the wing surfaces. Since the rear wing contains curved aerodynamic surfaces, narrow gaps, and endplate regions where complex vortices may occur, local mesh refinement was applied around the main plane, flap, flap gap, leading edge, trailing edge, and endplate edges. Boundary layer mesh elements were also added near the wing surface to improve the resolution of the near-wall flow [15]. A relatively coarser mesh was used in the far-field region to reduce the total number of elements and improve computational efficiency. Through this meshing strategy, the simulation could obtain sufficient accuracy in the key aerodynamic regions while maintaining reasonable computational cost.

Overall, the geometric model and computational domain were designed to provide a reliable basis for evaluating the aerodynamic performance of the Formula One rear wing [13]. The combination of simplified but representative geometry, appropriate boundary conditions, and locally refined mesh made it possible to analyze the downforce generation mechanism, drag characteristics, and flow-field behavior of the baseline rear wing model.

3.2. Governing Equations and Simulation Settings

The aerodynamic performance of the Formula One rear wing was simulated by solving the steady, three-dimensional, incompressible turbulent flow field. The governing equations include the continuity equation and the Navier--Stokes equations, which describe the conservation of mass and momentum, respectively.

The continuity equation is expressed as:

$$\nabla \cdot \mathbf{u} = 0 \tag{1}$$

where $\mathbf{u}$ denotes the velocity vector [6, 10]. This equation ensures that mass is conserved within the flow domain.

The momentum conservation is governed by the incompressible Navier--Stokes equations:

$$\rho(\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla p + \mu \nabla^2 \mathbf{u} \tag{2}$$

where ρ is the fluid density, p is the pressure, and μ is the dynamic viscosity [6, 10]. These equations describe the balance between inertial, pressure, and viscous forces acting on the fluid.

Due to the highly turbulent nature of the flow around the rear wing---especially near the endplates and in the wake region---a turbulence model is required to close the system of equations. In this study, the Shear Stress Transport (SST) k-ω turbulence model was employed. This model combines the advantages of the k-ω model in the near-wall region with the robustness of the k-ϵ model in the free stream, making it particularly suitable for predicting flow separation and adverse pressure gradients.

The transport equations for turbulent kinetic energy k and specific dissipation rate ω are solved together with the governing equations, enabling accurate prediction of boundary layer behavior and vortex structures [6].

For boundary condition settings, a uniform velocity inlet was applied at the upstream boundary to represent high-speed racing conditions. The outlet boundary was defined as a pressure outlet, allowing the flow to exit the computational domain smoothly. The surfaces of the rear wing and endplates were treated as no-slip walls. The ground was modeled as a moving wall with the same velocity as the incoming airflow to simulate realistic road conditions. Symmetry boundary conditions were applied where applicable to reduce computational cost.

The numerical solution was obtained using the finite volume method. Pressure--velocity coupling was handled using the SIMPLE algorithm, and second-order discretization schemes were adopted for both the momentum and turbulence equations to enhance numerical accuracy.

Convergence was assessed using both residual and physical criteria. The residuals of the governing equations were required to decrease below 10^{-5}, and the aerodynamic coefficients were monitored to ensure that they reached stable values [14]. Only when both residuals and force coefficients stabilized was the solution considered converged.

To evaluate the aerodynamic performance of the rear wing, the downforce coefficient C_L and drag coefficient C_D were defined as:

$$C_L = \frac{F_L}{\frac{1}{2}\rho V^2 A} \tag{3}$$

$$C_D = \frac{F_D}{\frac{1}{2}\rho V^2 A} \tag{4}$$

where F_L represents the downforce, F_D is the drag force, V is the freestream velocity, and A is the reference area.

The aerodynamic efficiency of the rear wing was evaluated using the lift-to-drag ratio:

$$\frac{C_L}{C_D} \tag{5}$$

A higher C_L indicates stronger downforce generation, while a lower C_D reflects reduced aerodynamic resistance [11]. The ratio C_L/C_D serves as a key indicator of aerodynamic efficiency, balancing cornering performance and straight-line speed.

These governing equations and simulation settings provide a rigorous and reliable framework for analyzing the aerodynamic characteristics of the rear wing and form the foundation for subsequent optimization studies [12].

4. Rear Wing Optimization and Results Discussion

4.1. Optimization Design of Rear Wing Parameters

To enhance the aerodynamic performance of the Formula One rear wing, a parametric optimization strategy was implemented based on the CFD model established in the previous section. The optimization focused on key geometric and aerodynamic variables that have a dominant influence on downforce generation and drag characteristics.

The primary optimization variables included the angle of attack (AoA), airfoil camber, flap gap, and endplate geometry. The angle of attack of both the main plane and the flap was adjusted to control the pressure distribution and flow deflection [15]. Increasing the AoA generally enhances downforce by intensifying the pressure difference between the upper and lower surfaces; however, excessive AoA may lead to flow separation and a rapid increase in drag. Therefore, AoA was treated as a sensitive variable requiring careful tuning.

Airfoil camber was modified to further improve lift (downforce) characteristics. A higher camber typically increases the curvature of the flow path, resulting in stronger acceleration over the upper surface and lower pressure regions. However, excessive camber may again induce early flow separation, especially under high-speed conditions. Thus, moderate camber enhancement was considered in the optimization process [10].

The flap gap, defined as the distance between the main plane and the upper flap, was also optimized. This parameter directly affects the interaction between multi-element wing components. Proper gap sizing allows high-energy flow to pass through the slot, re-energizing the boundary layer on the flap and delaying separation. Both overly small and excessively large gaps can reduce aerodynamic efficiency, making this parameter critical in multi-element wing design [6].

Endplate geometry was another important optimization variable. The shape and curvature of the endplates were adjusted to reduce spanwise flow and suppress wingtip vortices. By modifying the endplate contour and introducing smoother curvature transitions, the strength of the trailing vortices could be reduced, thereby decreasing induced drag and improving overall aerodynamic efficiency.

The optimization objective was defined as maximizing downforce while controlling the increase in drag. In practical terms, this corresponds to maximizing the lift coefficient C_L under the constraint that the drag coefficient C_D does not increase excessively [7]. Additionally, the lift-to-drag ratio C_L/C_D was used as a comprehensive performance indicator to evaluate the aerodynamic efficiency of different configurations.

To systematically evaluate the effectiveness of the optimization, two main configurations were defined: the baseline model and the optimized model. The baseline model represents the initial rear wing geometry before parameter adjustments, serving as a reference for comparison. The optimized model incorporates the improved parameter values obtained through iterative CFD simulations [15].

The parameter adjustment process followed a structured logic. First, a sensitivity analysis was conducted to identify the relative influence of each parameter on aerodynamic performance. Parameters with the most significant impact—such as AoA and flap gap—were prioritized in the early stages of optimization. Subsequently, a stepwise adjustment strategy was adopted, where each parameter was varied within a predefined range while keeping others constant, allowing for clear identification of its individual effect.

After identifying favorable parameter ranges, a combined optimization approach was applied. Multiple parameters were adjusted simultaneously to capture their interaction effects, which are particularly important in multi-element wing configurations. Iterative simulations were conducted until an optimal balance between increased downforce and controlled drag growth was achieved.

Through this systematic optimization design, the study establishes a clear framework for improving rear wing aerodynamic performance, providing a solid basis for the comparative analysis presented in the subsequent section.

4.2. Aerodynamic Performance Comparison and Flow Field Analysis

Based on the optimized rear wing parameters described in Section 4.1, a comparative analysis was conducted between the baseline model and the optimized model to evaluate aerodynamic performance improvements and underlying flow-field mechanisms.

First, the downforce performance was quantitatively compared. The optimized model exhibited a clear increase in downforce coefficient C_L relative to the baseline configuration. This improvement can be attributed to the enhanced pressure difference between the upper and lower surfaces of the rear wing. Specifically, the optimized angle of attack and camber increased flow acceleration over the suction surface, resulting in a larger low-pressure region. At the same time, the pressure on the pressure side remained relatively high, leading to a stronger net aerodynamic force acting downward. This increase in downforce directly contributes to improved tire grip, especially during high-speed cornering.

In terms of drag characteristics, the optimized model showed a moderate increase in drag coefficient C_D, which is expected due to the higher lift generation. However, the growth in drag was effectively controlled within an acceptable range [6, 10]. The improved endplate design and optimized flap gap helped reduce induced drag by weakening spanwise flow and limiting vortex intensity. As a result, although drag increased slightly, the overall aerodynamic efficiency, represented by the lift-to-drag ratio C_L/C_D, improved compared to the baseline model. This indicates that the optimization achieved a favorable balance between downforce enhancement and drag penalty.

The pressure contour analysis further explains these performance changes [6]. For the optimized model, a more pronounced low-pressure region was observed on the upper surface of both the main plane and the flap, while the lower surface maintained relatively higher pressure levels. The pressure distribution was also more uniform along the span, indicating improved flow attachment and reduced localized separation. In contrast, the baseline model showed weaker pressure gradients and some irregular pressure distribution near the wing tips, suggesting less efficient aerodynamic loading.

Velocity streamline analysis provided additional insight into the flow behavior. In the optimized configuration, the airflow remained more attached along the wing surfaces, with smoother acceleration over the upper surface and more coherent flow structures. The slot between the main plane and flap effectively guided high-energy flow onto the flap surface, delaying boundary layer separation. In the baseline model, partial flow separation and less organized streamlines were observed, particularly near the trailing edge and flap region, which reduced aerodynamic effectiveness.

The analysis of wake and vortex structures revealed significant differences between the two configurations [5, 8]. The optimized model exhibited weaker and more controlled wingtip vortices due to the refined endplate geometry. The reduction in vortex strength helped decrease induced drag and improve downstream flow stability. Additionally, the wake region behind the optimized wing showed a more streamlined structure with reduced turbulence intensity, which is beneficial for minimizing aerodynamic losses.

From a vehicle dynamics perspective, the improvements in aerodynamic performance have direct implications for racing performance. The increased downforce enhances tire-road contact, improving traction and stability, particularly in high-speed corners. Meanwhile, the controlled drag increase ensures that straight-line speed is not significantly compromised. Furthermore, the more stable and uniform flow field reduces aerodynamic fluctuations, contributing to better handling consistency and driver confidence [13].

Overall, the optimized rear wing configuration demonstrates superior aerodynamic characteristics compared to the baseline model. The combination of increased downforce, controlled drag growth, improved pressure distribution, and more stable flow structures confirms the effectiveness of the proposed optimization strategy. These results validate the importance of multi-parameter aerodynamic design in achieving high-performance Formula One rear wing configurations [11, 14].

5. Conclusion

This study systematically investigated the aerodynamic performance of a Formula One rear wing using computational fluid dynamics (CFD), with a focus on enhancing downforce while maintaining acceptable drag levels. A detailed geometric model and computational domain were established, and the governing equations of incompressible turbulent flow were solved using an appropriate turbulence model. Based on this framework, key rear wing parameters, including angle of attack, airfoil camber, flap gap, and endplate geometry, were selected for optimization, and a structured parametric design strategy was implemented.

The results demonstrate that the optimized rear wing configuration achieves a significant increase in downforce compared to the baseline model. This improvement is primarily attributed to the enhanced pressure distribution and more effective flow control over the wing surfaces. Although a moderate increase in drag was observed, it remained within a controlled range, leading to an overall improvement in aerodynamic efficiency as reflected by the lift-to-drag ratio. Flow-field analysis further confirmed that the optimized design promotes better flow attachment, reduces separation, and weakens vortex structures, thereby improving aerodynamic stability.

From a performance perspective, the increase in downforce directly contributes to improved vehicle stability and tire grip, particularly under high-speed cornering conditions. The optimized flow structures and reduced aerodynamic fluctuations also enhance handling consistency, which is critical for Formula One racing performance. These findings highlight the importance of multi-parameter rear wing optimization in achieving a balance between aerodynamic loading and efficiency.

However, several limitations remain in this study. First, the analysis was conducted on an isolated rear wing model, without considering the aerodynamic coupling effects with other vehicle components such as the front wing, diffuser, and car body. Second, the simulations were performed under steady-state conditions, which do not fully capture the transient aerodynamic behavior experienced during real racing scenarios. Third, the results were not validated against experimental data, such as wind tunnel measurements, which may affect the reliability of the conclusions.

Future research should address these limitations by incorporating full-vehicle aerodynamic coupling to better represent realistic flow interactions. Multi-objective optimization techniques can be applied to simultaneously balance multiple performance criteria, including downforce, drag, and flow stability. In addition, the integration of artificial intelligence and machine learning methods may significantly improve optimization efficiency and enable more advanced design exploration. Finally, experimental validation through wind tunnel testing or track data is necessary to further verify the accuracy and practical applicability of CFD-based aerodynamic optimization.

In summary, this study provides a comprehensive CFD-based framework for rear wing aerodynamic analysis and optimization, offering valuable insights into improving downforce and aerodynamic efficiency in Formula One racing applications.

References

1. G. Souza Dourado and M. T. Hayashi, "Optimization of a 2-D Multi-Element Rear Wing on a Formula 1 Car," in *SAE Brasil 2024 Congress*, Dec. 2024, 368540.
2. M. Ahsan, "Numerical Study on Aerodynamic Drag Reduction on a Rear Wing of a Formula Student Car," unpublished.

3. M. A. Dharmawan, D. D. P. Tjahjana, B. Kristiawan, and S. I. Pertiwi, "Design and aerodynamics analysis of rear wing formula student car using 3 dimension CFD (computational fluid dynamics)," in *AIP Conference Proceedings*, vol. 2217, no. 1, p. 030166, Apr. 2020, AIP Publishing LLC.
4. K. Chen and S. Liu, "Analysis and Optimization of Aerodynamic Performance of Race Car Rear Wing Based on CFD," *The Frontiers of Society, Science and Technology*, vol. 2, no. 11, pp. 104-116, 2020.
5. S. Wordley and J. Saunders, "Aerodynamics for Formula SAE: Initial design and performance prediction," SAE Technical Paper, no. 2006-01-0806, 2006.
6. R. O. Raj, N. Tharakaram, and N. Prakash, "Design and CFD analysis of a rear wing of a motorsport car," *International Journal of Vehicle Structures & Systems*, vol. 14, no. 1, pp. 58-62, 2022.
7. T. P. O'Driscoll and A. R. Barron, "CFD analysis of the location of a rear wing on an Aston Martin DB7 in order to optimize aerodynamics for motorsports," *Vehicles*, vol. 4, no. 2, pp. 608-620, 2022.
8. S. Durrer, "Aerodynamics of Race Car Wings: A CFD Study," unpublished, 2016.
9. N. M. Prieto Sánchez, "Design and Aerodynamic Analysis of a Formula Student Rear Wing," unpublished, 2023.
10. S. R. Hassan, T. Islam, M. Ali, and M. Q. Islam, "Numerical study on aerodynamic drag reduction of racing cars," *Procedia Engineering*, vol. 90, pp. 308-313, 2014.
11. V. Chiplunkar, R. Gujar, A. Adiverekar, R. Kulkarni, and A. Thonge, "Computational fluid dynamics analysis for an active rear-wing design to improve cornering speed for a high-performance car," *Materials Today: Proceedings*, vol. 77, pp. 887-896, 2023.
12. S. Gupta and K. Saxena, "Aerodynamics analysis of a formula sae car," unpublished, 2017.
13. C. Suvanjumrat, S. Namchanthra, T. Phengpom, J. Priyadumkol, W. Chookaew, S. Watechagit, et al., "AI-assisted CFD optimisation of multi-element wing angle of attack for enhanced formula SAE aerodynamic performance," *International Journal of Thermofluids*, 101440, 2025.
14. K. N. Nguyen, L. K. T. Vo, and T. D. Nguyen, "CFD Analysis of the Rear Wing Angle to Aerodynamic Characteristics on the Audi TT," *Mathematical Modelling of Engineering Problems*, vol. 12, no. 7, 2025.
15. G. Menchise and L. Ventre, "Multi-fidelity optimization procedure for a two-element Formula 1 rear wing," unpublished, 2021.