

Aerodynamic Design and CFD Simulation in Formula 1 Underbody Optimization

Haoran Sun *

Shanghai Nanyang Model Private School, Shanghai, 200232, China

* Corresponding Author Email: SAllenxxyw@outlook.com

Abstract. Since the modern era, aerodynamic design has become an indispensable element of automotive engineering, particularly in motorsport. Formula 1 stands as a prime example of aerodynamic design pushed to its absolute limits. Through meticulous aerodynamic configuration, Formula 1 cars generate immense downforce during races, thereby enhancing cornering speed and stability. In Formula 1, the underbody alone contributes more than half of the car's total downforce. Consequently, the aerodynamic design of the racing car's underbody plays a crucial role in its on-track performance. This article explores the principles of CFD simulation, including governing equations, turbulence models, and boundary condition settings. It examines diffuser design optimization and the degradation of aerodynamic performance under wake effects using CFD simulation methods. Furthermore, it discusses the limitations of CFD modelling and future directions for innovation in aerodynamic design.

Keywords: Aerodynamic Design, CFD Simulation, Downforce Generation, Underbody Optimization.

1. Introduction

As the epitome of extreme motorsport, the aerodynamic design of Formula 1 cars is the decisive factor in overall vehicle performance. Particularly crucial is the underbody design, which directly influences downforce generation, drag control, and the car's stability and handling characteristics. In Formula 1 racing, even the most subtle aerodynamic enhancements can yield significant lap time improvements. Consequently, the design and optimization of the underbody have remained a central focus for team engineers, particularly since the advent of the ground effect era in 2022. In recent years, the maturation of Computational Fluid Dynamics (CFD) technology has seen its application in aerodynamic research expand considerably, establishing it as the most effective tool alongside wind tunnel testing [1]. CFD enables precise simulation of airflow behavior in a virtual environment, allowing assessment of how different designs impact performance and thereby reducing development costs. However, challenges in F1 underbody design manifest in two key aspects: firstly, the underbody's position beneath the vehicle exposes it to complex interactions including ground effect, tyre turbulence, and the wake from preceding cars; secondly, regulatory constraints and the need to maintain underbody secrecy limit external visualization and modification, presenting significant research difficulties. Current aerodynamic research on F1 underbodies focuses on two key areas: firstly, employing CFD simulation to analyze the aerodynamic characteristics of components such as the diffuser, guide vanes, and winglets, including the detrimental effects of elements like vanes and rear wings on the wake of preceding vehicles; secondly, investigating the patterns of aerodynamic behavior and optimization strategies for underbodies under non-ideal flow conditions, such as those encountered in the wake of preceding cars [2]. Existing experiments demonstrate that optimizing diffuser structures enhances downforce while reducing drag [3]. Specifically, modifying the diffuser's cross-sectional shape can decrease drag to a certain extent while generating greater downforce. Laterally oriented guide vans can separate tyre-induced turbulence closer to the ground, increasing downforce by 13.49% and reducing drag by 5.58%. Rear flaps and underbody winglets can ameliorate flow separation and pressure distribution [4]. Nevertheless, current research exhibits several limitations: Firstly, most CFD simulations employ the Reynolds-Averaged Navier-Stokes (RANS) model, which offers computational efficiency but exhibits significant prediction errors for large-scale separated flows; Secondly, studies predominantly focus on steady-state conditions, failing to capture

transient flow structures and thus unable to accurately depict vortex formation and dissipation; Thirdly, due to confidentiality surrounding actual racing car models and data, most simulations employ simplified geometric models that cannot rival real vehicles, rendering precise and reliable predictions challenging [5].

Consequently, employing high-fidelity CFD methods, numerical simulations of the F1 car underbody were conducted using unstructured grids and boundary layer refinement techniques. Based on the incompressible Navier-Stokes equations and utilizing the SST $k-\omega$ turbulence model. This model is applicable for predicting reverse pressure gradients and separated flows. Boundary conditions including velocity inlet, pressure outlet, moving ground, and rotating tyre wall surfaces were employed to simulate real racing conditions. Mesh independence verification calculations were conducted to ensure precise and reliable results.

Through systematic analysis of how the modified underbody components function within the wake, the aerodynamic mechanisms of the underbody were investigated, providing reference for further design optimization. These findings contribute to enhancing the aerodynamic efficiency of the racing car while laying groundwork for dynamic condition studies and the development of active aerodynamics.

2. Method

This review summarizes computational fluid dynamics (CFD) methods used to investigate the performance impacts of Formula One chassis design. By solving fluid motion equations, CFD simulates real-world airflow phenomena in a virtual environment, making it the most effective tool for F1 aerodynamics research outside wind tunnels. By employing CFD, engineers can obtain highly intuitive representations such as pressure contour plots, vortex streamlines, and component load distributions, thereby facilitating the analysis of aerodynamic interactions between various assemblies.

2.1. Principles

The essence of CFD simulation lies in solving the Navier–Stokes equations (the conservation laws governing fluid motion) under the assumption of incompressible fluids. This system of equations primarily includes:

$$\frac{\partial \rho^*}{\partial t^*} + \nabla^* \cdot (\rho^* v^*) = 0 \quad (1)$$

$$\left(\frac{\partial v^*}{\partial t^*} + v^* \cdot \nabla^* v^* \right) = -\nabla^* p^* + \frac{1}{Re} \nabla^{*2} v^* + f^* \quad (2)$$

Where ρ^* is the non-dimensional density of air, v^* is the non-dimensional velocity field, p^* is the non-dimensional pressure, f^* is the non-dimensional external body forces acting on the fluid, Re is the Reynolds number $\left(Re = \frac{\rho V L}{\mu} \right)$, ρ is the density of air, V is the velocity of fluid, L is the characteristic length, and μ is the dynamic viscosity of the fluid [4].

2.2. Turbulence Models

The flow around Formula 1 race cars exhibits high Reynolds number turbulence. Directly employing Direct Numerical Simulation (DNS) is prohibitively expensive, so Reynolds-Averaged Navier-Stokes (RANS) methods are universally adopted. Within the RANS framework, the Shear Stress Transport ($k-\omega$) model has become the preferred choice for studying under body airflow due to its superior prediction capabilities for reverse pressure gradients and separated flows. This model is also employed in all papers covered by this review. It utilizes the $k-\omega$ model in the near-wall region and transitions to the $k-\epsilon$ model in the far-field region, thereby combining the advantages of both for simulation [2].

2.3. Computational Setup and Boundary Conditions

2.3.1. Computational Domain

As shown in Fig. 1, a rectangular virtual wind tunnel significantly larger than the F1 car is employed to eliminate boundary effects.

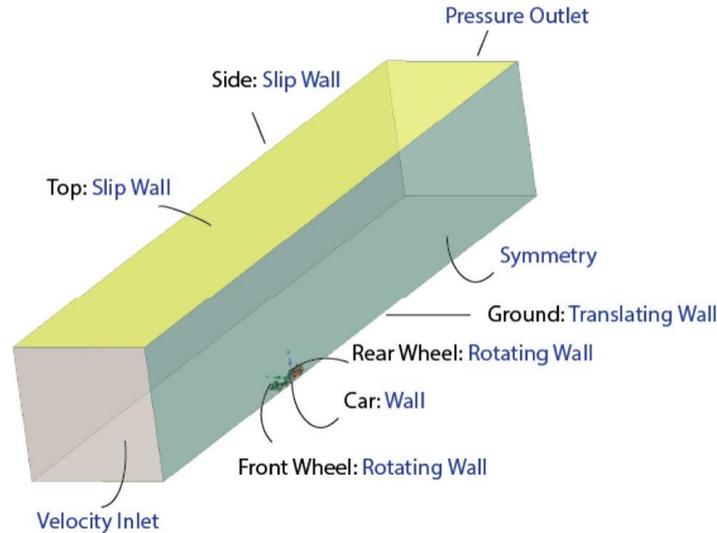


Figure 1. Computational domain with boundary names (in black) and conditions (in blue) [6]

2.3.2. Mesh

A non-structured mesh was employed, with local refinement applied near the body surface, bottom plate, and regions with anticipated high flow gradients. Furthermore, to accurately resolve the boundary layer, the height of the first mesh layer was set such that the non-dimensional wall distance (y^+) was less than 1 from the steel wall surface, using a multi-layer prismatic inflation mesh.

$$\delta = \frac{5x}{\sqrt{Re}} \quad (3)$$

$$Re = \frac{\rho VL}{\mu} \quad (4)$$

Where δ is the thickness of the first cell layer near the wall in the mesh, x is the distance from the wall to the first cell layer, re is the Reynolds number calculated using Eq. (4), ρ is the air density, V is the fluid velocity, L is the characteristic length, and μ is the dynamic viscosity of the fluid.

In the research paper “*Investigating the Impact of Structural Features on the Performance of F1 Race Car Rear Wings Using CFD (Computational Fluid Dynamics) Methods,*” the focus is on the grid independence study. During the numerical simulations, an 8.25-million-cell grid model was employed (as shown in Table 1). Based on the required computational accuracy and available computational resources, the total number of grid cells was determined to be 8.25 million, ensuring that the simulation met the desired precision standards (as shown in Fig. 2) [4].

Table 1. The sensitivity of the diffuser to mesh size

Base Size (mm)	Total Elements	Drag Force (N)	Computational Time (Hour)
10	1.72 M	177	1
7	3.25 M	182	1.5
5	5.6 M	195	3.3
4	8.25 M	197	5
3	11 M	197	10.2

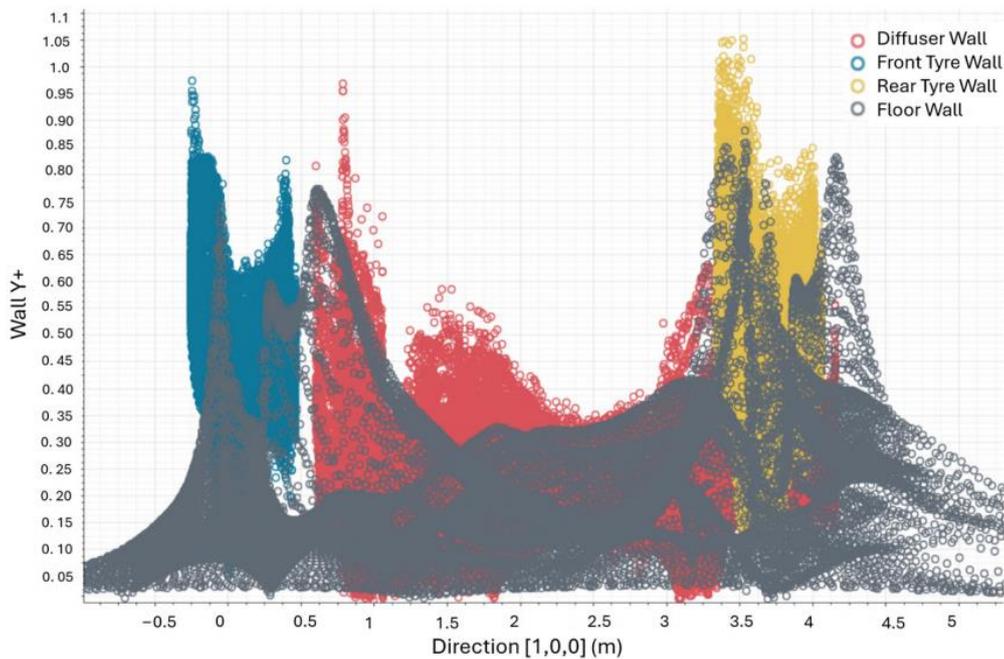


Figure 2. Distribution of Y^+ along the length of diffuser [4]

2.3.3. Boundary Conditions

Inlet: The inlet boundary condition specifies the incoming flow velocity.

Outlet: The outlet boundary condition specifies static pressure.

Ground: Defined as a moving wall with a velocity equal to the incoming flow velocity, in order to simulate the relative tangential motion of the vehicle during racing.

Tires: The original tire surfaces are replaced with rotating wall boundaries, with their respective angular velocities specified.

Body Surface: Defined as a no-slip wall boundary (as shown in Table 2 and 3).

Table 2. Setup of the simulation

Variable	Value
Free stream velocity	50 m/s
Fluid density	1.225 kg/m ³
Turbulent Intensity	0.15%
Turbulent length scale	3.475 m
Reynolds Number	12 x 10 ⁶

Table 3. Boundary conditions for cases

Case Number	Inlet Velocity	Ride Height	Intake/Exhaust Mass Flow Rate (kg/s)
1	30	30	0.132
2	30	15	0.132
3	50	30	0.22
4	50	10	0.22
5	90	30	0.4
6	90	5	0.4

3. Results

3.1. Improvements and Optimization of the Diffuser Component Design

The diffuser generates downforce through three distinct mechanisms: *diffuser pumping*, *diffuser upswEEP*, and *diffuser-ground interaction* [7]. Eugeni Pérez Nebot’s study examined the aerodynamic performance effects resulting from structural modifications to the diffuser [4].

Ground effect is one of the primary mechanisms responsible for producing downforce in racing aerodynamics. The performance of a diffuser is primarily influenced by its geometric parameters—specifically, its angle of attack and ground clearance. As the ground clearance decreases, the downforce produced by the diffuser increases progressively. However, once a critical ride height is reached, the diffuser experiences *diffuser stall*, resulting in a sudden loss of downforce. A larger diffuser angle of attack makes the system more prone to stalling [4].

As illustrated in Fig. 3, a 10° diffuser begins to stall at $h/L = 0.02$, whereas a 15° diffuser stalls as early as $h/L = 0.022$. The magnitude of the maximum negative pressure generated at the diffuser inlet determines the overall downforce level. Furthermore, altering the fore–aft position of the diffuser can shift the vehicle’s pressure center, thereby affecting its aerodynamic balance and stability [5].

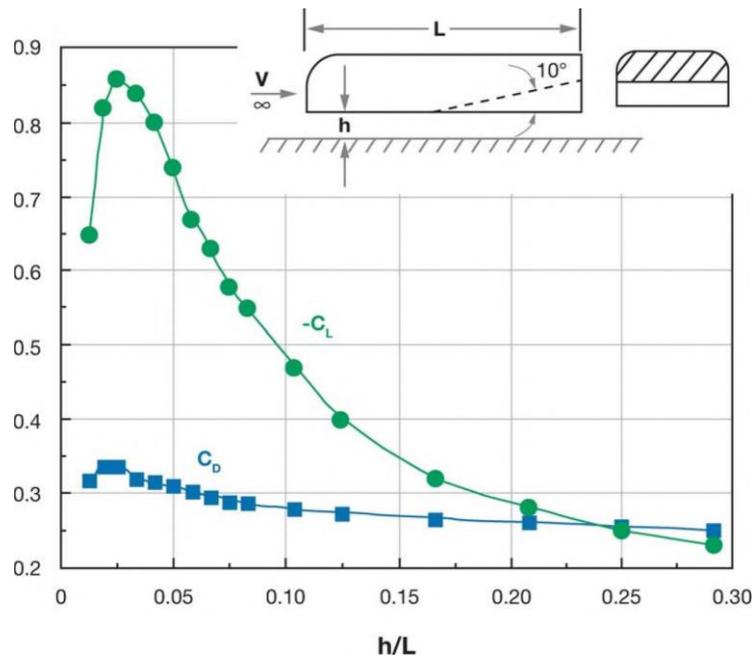


Figure 3. Lift and drag coefficient variation with ground clearance for a generic model with underbody diffuser [5]

3.1.1. Lateral Vanes

Positioning deflector blades on the inner side of the diffuser’s diverging section separates the turbulent airflow generated by the tires from the primary diffuser flow path. This redirection guides the turbulent flow beneath the central underbody airflow, thereby reducing energy losses. Consequently, the underbody downforce increased by 13.49%, while the drag decreased by 5.58% (as shown in Table 4). These results demonstrate that optimized vortex management enhances downforce and aerodynamic efficiency while simultaneously reducing drag [4].

Table 4. The enhancement effect of installing lateral vanes

Model	Downforce on Floor (N)	Improvement %	Drag Force on Floor (N)	Improvement %
Baseline	-2334.100	-	43.93	-
Case Study 1	-2649.063	13.49	41.475	5.58
Model	Downforce on R-Tire (N)	Increment %	Drag Force on F-Tire (N)	Change %
Baseline	237.17	-	211.4	-
Case Study 1	253.92	6.75	211.9	0.23

3.1.2. End Flaps

By installing end flaps with a 10cm chord length at the diffuser's trailing edge and adjusting their height, different effects can be achieved. Such flaps can increase down force by approximately 21%, but simultaneously cause the drag coefficient to rise by about 75% (As shown in Table 5).

Table 5. The effect of rear diffuser height variation on underbody aerodynamic performance

Model	Downforce on Floor (N)	Improvement %	Drag Force on Floor (N)	Improvement %
Baseline	-2334.100	-	43.93	-
Case Study 2	-2843.690	21.833	77.441	76.282
Case Study 2.2	-2813.304	20.53	75.348	71.501
Case Study 2.3	-2822.502	20.93	72.133	64.199

3.1.3. Underbody Flaps

Positioning the flaps beneath the diffuser prevents them from being directly exposed to high-speed airflow, achieving a 6.6% increase in downforce and a 44.45% reduction in drag (As shown in Table 6). This configuration enhances the effectiveness of flow separation at the diffuser outlet, thereby significantly reducing the scale of the low-pressure wake region [4].

Table 6. The aerodynamic performance of floor on adding flaps underneath the diffuser

Model	Downforce on Floor (N)	Improvement %	Drag Force on Floor (N)	Improvement %
Baseline	-2334.100	-	43.93	-
Case Study 3	-2488.268	6.6	24.403	44.45

3.1.4. Deflector

In *Formula 1 Race Car Aerodynamics*, advanced Line Integral Convolution (LIC) and Volume Rendering techniques were employed to integrate streamline and bubble trace images into the flow visualization. Based on these methods, the function of the deflector plate was investigated. Experimental results revealed that the deflector generates significant vortex structures, which prevent high-pressure airflow from the vehicle’s sides from entering the low-pressure region beneath the underbody, thereby maximizing ground-effect retention.

Furthermore, the influence of ride height must be considered when designing the deflector. Lower ground clearance promotes stronger ground-effect generation; however, excessive reduction can cause flow separation. This necessitates a highly smooth underbody surface and the optimization of ground clearance to achieve a balanced combination of aerodynamic efficiency, stability, and control capability [6].

3.1.5. Vortex Generators

Vortex generators (VGs) are among the most effective and widely used devices for enhancing underbody downforce in racing vehicles. Their operating principle involves mounting a series of vertical blades with varying shapes, angles, and lengths at the front section of the underbody. As the airflow moves along the vehicle’s longitudinal axis and encounters these blades, stable vortices are generated within the airstream. These vortices are subsequently converted rearwards, where they interact with the underbody and diffuser surfaces to create low-pressure regions. As vehicle speed increases, the resulting downforce on the bodywork correspondingly rises.

Experimental data indicate that the intensity of vortices produced by VGs increases as ground clearance decreases (as shown in Fig. 4). Furthermore, as the ground clearance is reduced, the vortices move closer to the underbody, thereby strengthening the suction effect [5]. However, when the clearance becomes excessively low, the vortices may break down, leading to vortex instability and flow separation. Consequently, a further reduction in ground clearance can paradoxically reduce downforce and impair the vehicle’s aerodynamic stability. Therefore, VG design requires a careful balance between vortex strength and stability [5].

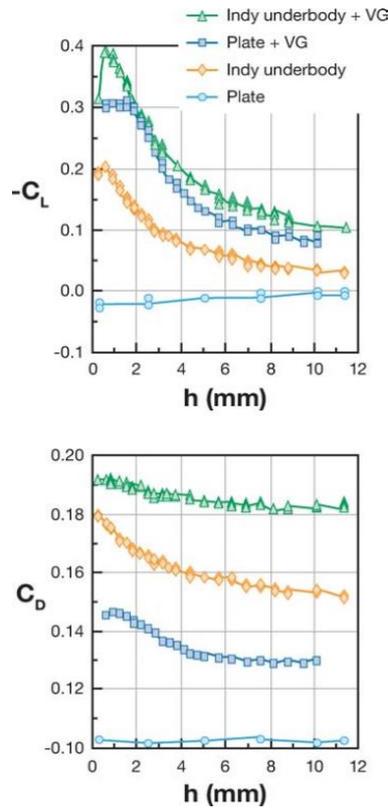


Figure 4. Downforce and drag coefficient variation with ground clearance for the generic Indy car underbody [5]

3.2. Impact of Wake Effects

3.2.1. Performance Degradation Quantification

As shown in Fig. 5 and 6, as the following distance (D) decreases, the aerodynamic performance of the trailing car deteriorates significantly. At $D = 2L$ (where L represents the car length), the trailing car loses approximately 23.5% of its total downforce. When D is reduced to $0.25L$, this loss increases to 62%. Although drag also decreases substantially by up to 40%. This reduction is not entirely advantageous for race car performance, as it results from a loss of kinetic energy in the upstream flow [2].

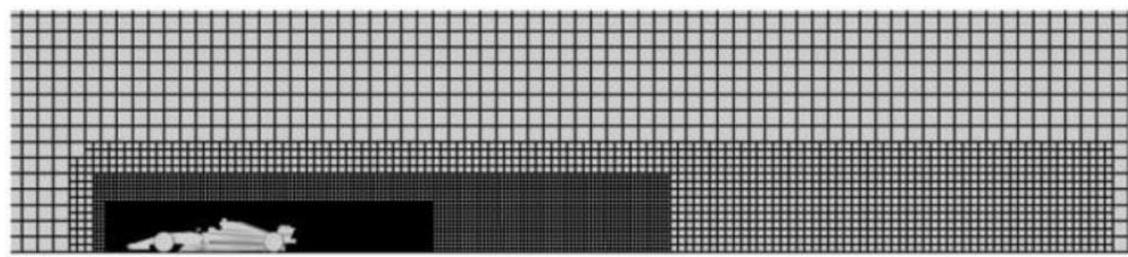


Figure 5. Overall mesh and refinement enclosures of the free stream case [2]

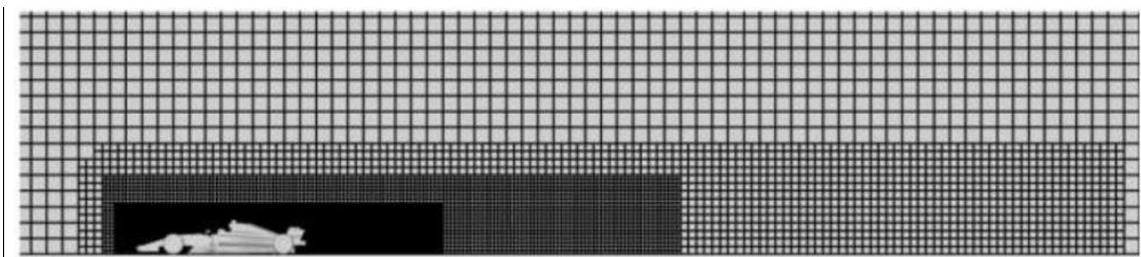


Figure 6. Overall mesh and refinement enclosures under wake effects [2]

3.2.2. Component Sensitivity Differences

Additionally, the study examined aerodynamic loss patterns across different vehicle components. The diffuser experienced the most severe performance degradation, with downforce decreasing by 70.2% at $D = 0.25L$. The rear wing began to exhibit performance decline at larger following distances and showed an approximately linear decay trend, whereas the front wing displayed noticeable losses only when $D < 1L$ [2].

Beyond the diffuser, the paper “*Aerodynamics of a Wing in Ground Effect in Generic Racing Car Wake Flows*” also reported that “*when subjected to dirty air conditions, the loss of lift in the mid-span region of the wing is more pronounced than in the wing-tip region.*” This observation further supports the existence of sensitivity differences among aerodynamic components [8].

3.2.3. Aerodynamic Balance Disruption

Another critical finding concerns the substantial shift in front-axle downforce contribution, commonly referred to as *front balance (FB)*. In the wake of a leading car, the rear diffuser and rear wing experience much greater downforce losses than the front wing, causing the aerodynamic center to move forward and increase the FB value by more than 28% [2].

This phenomenon has also been confirmed experimentally in a separate study. The original paper noted that, under moving-ground conditions, wind tunnel tests could reproduce a maximum vehicle spacing of approximately one car length, representing the typical distance maintained before an overtaking maneuver. Using identical spacing conditions, the study conducted computational fluid dynamics (CFD) simulations that employed both a wake-generator model and a complete rear-vehicle model. The results revealed that, compared with an isolated single-vehicle condition, both the aerodynamic drag and total downforce of the trailing vehicle decreased. Simultaneously, its front-axle aerodynamic balance dropped significantly, with reductions reaching up to 11% [9].

This front-balance reduction induces pronounced oversteer, making it difficult for drivers to maintain control through corners. It also explains why drivers often commit seemingly inexplicable errors when closely following another car on track [10].

4. Summary

The combined application of aerodynamic devices offers an effective means of simultaneously reducing overall vehicle drag while increasing downforce. CFD simulation results confirm that integrating a rear spoiler with a diffuser yields significant drag reduction across all operating conditions, with only a minimal influence on downforce generation. In contrast, employing a rear wing alone can generate substantial downforce and achieve even negative lift but also increases drag by 38.47%.

This configuration is particularly suitable for closed-circuit racing environments where downforce demands outweigh top-speed requirements. In such cases, prioritizing downforce generation is advantageous. For practical implementation, a comprehensive comparative analysis should be conducted to evaluate the drag-reduction and downforce-enhancement effects of each configuration, considering circuit-specific characteristics such as long straights and high-curvature sections.

First, computational accuracy is often constrained by the choice of turbulence model, with RANS models potentially exhibiting biases in predicting large-scale separated flows. Second, most studies still employ steady-state simulations, failing to capture true transient flow structures. Finally, constrained by high computational costs, geometric models are predominantly simplified versions or publicly available designs from previous seasons, presenting significant discrepancies compared to the complex, highly confidential designs actually used by real-world teams.

Future research will advance toward higher fidelity and greater integration. First, employing Large Eddy Simulation (LES) and high-accuracy hybrid simulation methods to capture more realistic turbulent structures. Second, conducting dynamic condition studies, such as analyzing the instantaneous aerodynamic performance of underbody components during bumpy road surfaces,

cornering, or lane changes. Third, integrate artificial intelligence (AI) and machine learning (ML) with CFD for rapid automated shape optimization and flow field feature recognition, enhancing research efficiency. Fourth, develop active aerodynamic underbody concepts capable of adjusting flap angles to alter geometry based on real-time conditions (straight-line, cornering, following), balancing peak performance with operational robustness.

References

- [1] Toet W. Aerodynamics and aerodynamic research in Formula 1. *The Aeronautical Journal*, 2013, 117 (1187): 1 – 26.
- [2] Guerrero A, Castilla R. Aerodynamic study of the wake effects on a Formula 1 car. *Energies*, 2020, 13 (19): 5183.
- [3] Katz J. Aerodynamics in motorsports. *Proceedings of the Institution of Mechanical Engineers, Part P: Journal of Sports Engineering and Technology*, 2021, 235 (4): 324 – 338.
- [4] Nebot E P, Gupta A, Mahak M. Investigating the impact of structural features on F1 car diffuser performance using computational fluid dynamics (CFD). *Mathematics*, 2025, 13 (9): 1455.
- [5] Katz J. Aerodynamics of race cars. *Annual Review of Fluid Mechanics*, 2006, 38 (1): 27 – 63.
- [6] Shaalan A, Assanis D, Raman A, Wijeyakulasuriya S, Senecal K. Formula 1 race car aerodynamics: Understanding floor flow structures and why it is a key component in modern racing. *SAE Technical Paper*, 2024, No. 2024 – 01 - 2078.
- [7] Ehirim O, Knowles K, Saddington A. A review of ground-effect diffuser aerodynamics. *Journal of Fluids Engineering*, 2018, 141 (2): 020801.
- [8] Soso M, Wilson P. Aerodynamics of a wing in ground effect in generic racing car wake flows. *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, 2006, 220 (1): 1 – 13.
- [9] Newbon J, Sims-Williams D, Dominy R. Aerodynamic analysis of Grand Prix cars operating in wake flows. *SAE International Journal of Passenger Cars – Mechanical Systems*, 2017, 10 (2): 318 – 329.
- [10] Nath D S, Pujari P C, Jain A, et al. Drag reduction by application of aerodynamic devices in a race car. *Advances in Aerodynamics*, 2021, 3 (4): 4.