Review Article

Reducing Drag by Optimizing the Underbody with Ride Height in Formula 1

Varun Pothamsetti

Ridge High School, 268 S Finley Ave, Basking Ridge, NJ, 07920, USA

ABSTRACT

This study investigates the aerodynamic interaction between wheel wakes and the underbody of Formula 1 cars, focusing on the effect of varying ride heights on drag reduction and ground effect optimization. With the reintroduction of ground effect principles in the 2022-2026 Formula 1 regulations, the potential for enhanced vehicle performance through improved aerodynamic design is significant. However, the efficiency of venturi tunnels, essential for generating effective downforce, is compromised by turbulent airflows produced by the wheels, known as wheel wakes. This research utilizes computational fluid dynamics (CFD) simulations to model these interactions at different ride heights, aiming to pinpoint optimal configurations that minimize aerodynamic drag while maximizing downforce. Initial findings suggest a delicate balance between ride height adjustment and tunnel geometry optimization, offering potential pathways to achieve aerodynamic efficiency in modern Formula 1 vehicles. This paper contributes to the evolving discourse on high-speed vehicle aerodynamics, providing insights that could inform future vehicle design and regulatory frameworks in motorsports.

Keywords: ANSYS; CFD; Formula One; Formula 1; Computational Fluid Dynamics; Navier Stokes

INTRODUCTION

In Formula 1 racing, the 2022-2026 regulations have brought back the ground effect, a key aerodynamic concept banned since the early 1980s. Ground effect works by shaping the car's underbody to create a low-pressure zone, which increases the car's grip on the track through aerodynamic downforce. This helps improve cornering speeds without relying solely on mechanical grip (1).

This research paper delves into the dynamic interaction between wheel wake and the car underbody as influenced by variations in ride height. The primary objective is to elucidate how changes in ride height affect this interaction and to identify strategies for mitigating drag through aerodynamic optimization. By employing computational fluid dynamics (CFD) simulations, this study seeks to model these interactions and pinpoint optimal configurations that minimize aerodynamic drag while maximizing downforce.

Historically, ground effect was first introduced in the late 1970s. These early ground effect cars used skirts -

---

Corresponding author: Varun Pothamsetti, E-mail: pothamsettivarun@gmail.com.

Received August 31, 2024**; Accepted** October 28, 2024
https://doi.org/10.70251/HYJR2348.242933

literal physical barriers that extended from the car's body to the ground to seal the underbody and maximize downforce by preventing high-pressure air from entering the low-pressure zone under the car. This significantly enhanced downforce and cornering speeds. However, the unpredictability and potential instability of these cars, combined with safety concerns, led to the banning of ground effect in 1983 (2).

The reintroduction of ground effect in 2022 aims to leverage advancements in technology and aerodynamics to harness these benefits while maintaining high safety standards. Unlike the skirts used in the late 1970s, modern F1 cars achieve the necessary sealing effect using air. This involves designing the floor and underbody aerodynamics in such a way that vortices generated by the car itself effectively seal the floor to prevent high-pressure air from disrupting the low-pressure zone. This sophisticated use of airflow management allows modern F1 cars to benefit from ground effect while adhering to safety and regulatory constraints.

The current regulations have reinstated venturi tunnels—underbody channels designed to accelerate airflow, thereby maximizing the ground effect. However, the proximity of the car's wheels to these tunnels introduces complexities due to wheel wake, which is a turbulent airflow pattern generated by the wheels. This turbulence can disrupt the streamlined air necessary for optimal ground effect, significantly degrading the efficiency of venturi tunnels and increasing aerodynamic drag.

METHODS AND MATERIALS

Model Development and Simulation Setup

A scale model of a 2023 F1 car was provided by Virtual Racing Cars LLC, incorporating detailed geometry of the chassis, floor, and venturi tunnels (Figure 1). The CFD software was set up to simulate airflow at speeds typical of F1 racing (300 km/h). 3 varying ride heights of 1 mm, 30 mm (typical of an F1 car), and 130 mm were tested.

Geometry and Domain Setup

The first step involved importing the geometry of the F1 car model into Ansys Discovery. After successfully importing the model, an external flow simulation was selected to analyze the behavior of the airflow around the car. This process automatically generated a fluid volume surrounding the model, defining the computational domain. The enclosure represents the region of interest where the airflow interactions with the car body were studied.

The fluid domain was bounded by an inlet, facing the front of the car, and a grounding plane to represent the track surface. The inlet boundary was critical in directing the airflow towards the car, mimicking real-world conditions on a racetrack. The grounding plane was set to simulate the effects of the car's proximity to the track, a crucial factor in understanding ground effect aerodynamics.

Boundary Conditions and Fluid Properties

The boundary conditions were meticulously defined to replicate real-world scenarios. The working fluid was set to gaseous air, a standard assumption for such simulations due to its well-known properties. The velocity at the inlet was defined as 300 km/h (83.33 m/s), which corresponds to typical speeds encountered in F1 races. This velocity was crucial for analyzing the high-speed aerodynamic phenomena around the vehicle.

The outlet boundary condition was set to a pressure outlet, with a gauge pressure of 0 Pa, allowing the airflow to exit the domain naturally. The grounding plane was treated as a no-slip wall, ensuring that the velocity of the airflow at the car's contact points with the ground was zero, accurately reflecting the real-world scenario.

Governing Equations

The simulations were governed by the Navier-Stokes equations, which describe the motion of fluid substances by accounting for the conservation of mass, momentum, and energy within the fluid domain. In the context of this study, the primary focus was on the conservation of momentum, which is crucial for understanding the aerodynamic forces acting on the car.

For incompressible flow, which is a valid assumption at the speeds typical of Formula 1 racing, the momentum

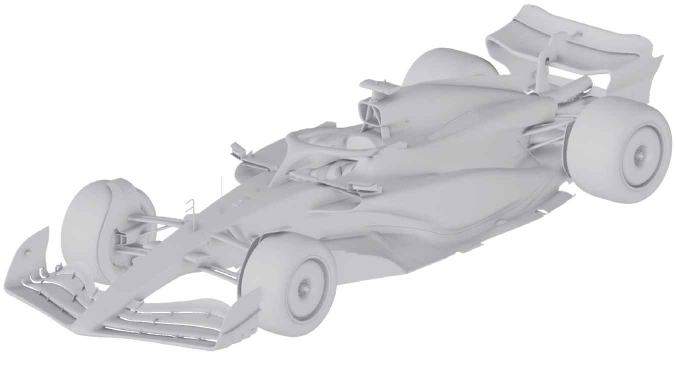

Figure 1. The Virtual Racing Cars Formula Alpha 2023.

equation of the Navier-Stokes equations is expressed as:

$$\frac{\partial u}{\partial t} + (u \cdot \nabla)\, u = \nu \nabla^2 u - \frac{1}{\rho} \nabla p + f$$

Where:

u is the velocity field of the fluid,

ρ is the fluid density,

p is the pressure field,

$\nu = \mu / \rho$ is the kinematic viscosity,

f represents external forces per unit mass (such as gravity).

This equation describes how the velocity of the fluid changes in response to the forces acting on it.

In this simulation, certain assumptions were made to simplify the analysis:

- Incompressible Flow Assumption: The flow was treated as incompressible, meaning that the fluid density ρ was assumed to be constant throughout the domain. This assumption holds for the speeds encountered in F1 racing, where compressibility effects are minimal.
- Steady-State Conditions: The simulations were conducted under steady-state conditions, meaning that the velocity field was time-independent ($\partial u/\partial t = 0$). This approach focuses on the equilibrium state of the airflow around the car, which is crucial for understanding the aerodynamic forces in a stable racing scenario.
- Neglecting External Forces: The term f, representing external forces such as gravity, was neglected in this analysis. The focus was on the aerodynamic forces generated by the interaction between the airflow and the car's surface.

With these assumptions, the momentum equation simplifies to:

$$(u \cdot \nabla)\, u = \nu \nabla^2 u - \frac{1}{\rho} \nabla p$$

This simplified equation was solved using the finite volume method, a numerical technique that discretizes the fluid domain into small control volumes. The CFD software then calculated the flow properties at each control volume, providing detailed insights into the airflow patterns around the car.

Data Collection and Expected Results

The velocity of the air in and around the venturi tunnels will be recorded. Special attention will be paid to the areas immediately following the wheel to assess the extent of wake interference. The results are anticipated to reveal a complex relationship between ride height and airflow disruption caused by wheel wakes. Lower ride heights are hypothesized to exacerbate wake interference due to closer proximity to the venturi tunnels, potentially increasing drag. Conversely, higher ride heights may demonstrate reduced interference but could also diminish the overall ground effect. Optimal configurations likely exist that balance these factors, offering a reduced drag coefficient while maintaining effective downforce generation.

RESULTS & DISCUSSION

Influence of Ride Height on Air Flow

The car's underbody spans approximately 3340 mm from the venturi tunnels' entrance to the diffuser's exit while spanning approximately 1600 mm in width. The height of the floor varies, with a significant constriction in the middle to enhance the venturi effect. To understand the impact of ride height on the performance of the venturi tunnels, we can use a dimensionless Aspect Ratio (AR), defined as the ratio of the product of the length and width of the floor to the ride height: $AR = (L \times W) / H$, where L is the length of the floor (3340 mm), W is the width of the floor (1600 mm), and H is the ride height.

At a lower ride height (1 mm): $AR_{low} = (3340 \text{ mm} \times 1600 \text{ mm}) / 1 \text{ mm} \approx 5{,}344{,}000$. The proximity of the car's underbody to the ground enhances the venturi effect, significantly accelerating the airflow under the car and creating strong downforce. However, this high aspect ratio indicates that the airflow is highly constrained, increasing the potential for turbulent wakes from the wheels to interfere with the streamlined flow in the venturi tunnels.

At a medium ride height (30 mm): $AR_{med} = (3340 \text{ mm} \times 1600 \text{ mm}) / 30 \text{ mm} \approx 178{,}133$. With a slightly lower aspect ratio, there is more room for the air to flow between the ground and the car. This reduces the intensity of the interference from the wheel wakes, balancing downforce and drag reduction.

At a higher ride height (130 mm): $AR_{high} = (3340 \text{ mm} \times 1600 \text{ mm}) / 130 \text{ mm} \approx 41{,}108$. The flow becomes less constrained by the ground with a lower aspect ratio. This potentially reduces the effectiveness of the ground effect due to the decreased influence of the venturi tunnels. The wheel wakes have more space to dissipate, which might reduce their disruptive effect but at the cost of lowering overall downforce (Figure 2 and Figure 3).

By using this modified Aspect Ratio, we can quantitatively compare how different ride heights influence the aerodynamic efficiency of the venturi tunnels. This approach provides a clearer understanding of the trade-offs involved in setting the ride height for optimal performance.

Drag Reduction, Downforce Optimization, and Strategic Adjustments

The optimal ride height seems to be a trade-off between minimizing the impact of wheel wakes (which disrupts airflow and increases drag) and maximizing the venturi effect (which enhances downforce). The medium ride height might offer the best compromise, as it allows enough airflow to minimize turbulence from the wheels while still maintaining effective downforce generation.

Aerodynamic modifications such as adjusting the geometry and positioning of venturi tunnels, alongside careful tuning of the ride height, can help in managing the turbulent airflows around the wheels and enhance the car's aerodynamic performance. Optimizing these factors is crucial for achieving a balance between speed and stability, particularly in high-speed corners.

Comparative Analysis

Zhang et al. [2006] emphasizes the critical role of ground effect aerodynamics in generating downforce through low pressure on surfaces nearest to the ground. This foundational understanding underscores the importance of optimizing the design and configuration of venturi tunnels to maximize downforce while minimizing drag. Katz highlights those components operating in close proximity to the ground, such as wings and diffusers, are highly efficient in contributing to downforce with minimal drag. This aligns with the current study's focus on optimizing the venturi tunnels' configuration to enhance downforce (3).

Diasinos et al. [2017] investigate the impact of varying ride heights on the aerodynamic interactions between the front wing and wheel of a racing car. Their findings indicate that different height-to-chord ratios significantly influence the vortex flow patterns and the resulting aerodynamic forces. Particularly, their study shows that lower ground clearance can exacerbate the destruction of the main vortex in the wheel contact patch, which directly affects downforce and drag. This complements our findings that lower ride heights can increase turbulence and disrupt the streamlined airflow, potentially increasing drag while enhancing downforce (4).

Mokhtar [2008] provides insights into how small changes in ground clearance can significantly alter downforce generation, especially with high-lift airfoil sections. Bevilaqua's research suggests that at very low clearances, the increased interaction with the ground

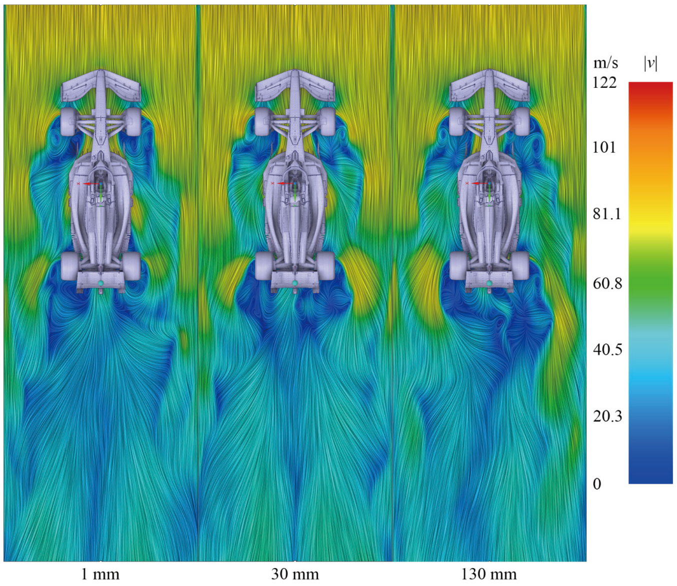

Figure 2. Top View Airflow Patterns at Different Ride Heights. This figure shows the flow of air on the ground around the Formula 1 car model at ride heights of 1 mm, 30 mm, and 130 mm. Color gradient represents the magnitude of the velocity of air.

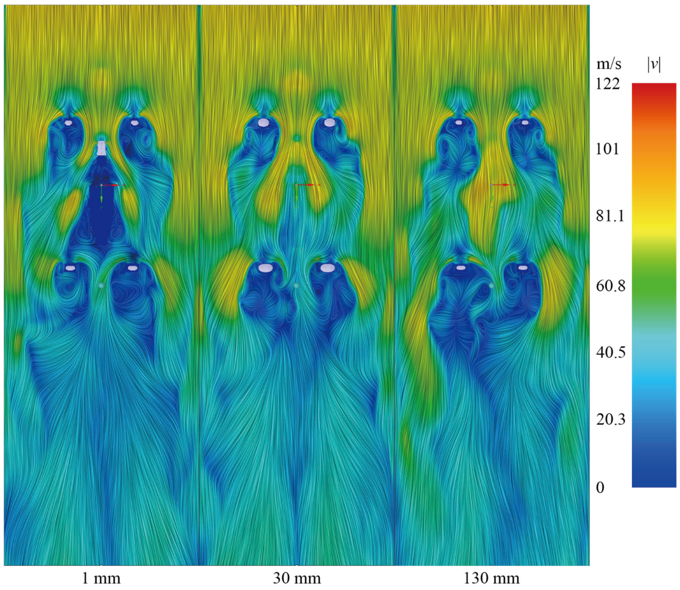

Figure 3. Bottom view showing the flow of air on the ground around the Formula 1 car model at ride heights of 1 mm, 30 mm, and 130 mm. Color gradient represents the magnitude of the velocity of air.

boundary layer may lead to a decrease in downforce despite the high-lift design. This phenomenon was also observed in our study, where extremely low ride heights, while increasing downforce, also posed the risk of reduced efficiency due to turbulent interactions (5).

CONCLUSION

The findings of this study suggest that there is a delicate balance required in adjusting ride height to optimize both drag reduction and downforce. Medium ride heights might offer a viable solution by providing a balance between reduced wheel wake interference and effective utilization of ground effects. These insights are pivotal for teams aiming to enhance vehicle performance under the current Formula 1 regulations that emphasize ground effect aerodynamics. Further advancements in CFD techniques could provide more detailed insights into the complex interactions between wheel wakes and underbody aerodynamics at various ride heights. Integrating more sophisticated models that account for impermanent effects and real-time adjustments could enhance the accuracy of predictions. Furthermore, while CFD simulations offer valuable insights, experimental validation through wind tunnel testing and on-track measurements would be crucial for confirming the theoretical findings. These experiments could help identify practical limitations and optimize configurations under real-world conditions.

DECLARATION OF CONFLICT OF INTERESTS

The author declares that there are no conflicts of interest regarding the publication of this article.

REFERENCES

1. 2022 FORMULA 1 TECHNICAL REGULATIONS. (2022, February 18). https://www.fia.com/sites/default/files/formula_1_-_technical_regulations_-_2022_-_iss_9_-_2022-02-18.pdf (pp. 16-21) (accessed on 2023-10-20).
2. Somerfield M. (2017, February 19). Retro F1 tech: The ground effect era. Motorsport.com. https://us.motorsport.com/f1/news/retro-f1-tech-the-ground-effect-era-873918/3014985/ (accessed on 2023-10-20).
3. Zhang X, Toet W & Zerihan J. (2006). Ground Effect Aerodynamics of Race Cars. Applied Mechanics Reviews - APPL MECH REV, 59. doi:10.1115/1.2110263 (accessed on 2024-5-18).
4. Diasinos S, Barber T & Doig G. Numerical analysis of the effect of the change in the ride height on the aerodynamic front wing–wheel interactions of a racing car. Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering. 2017; 231(7): 900–914. doi:10.1177/0954407017700372 (accessed on 2024-5-18).
5. Mokhtar W. (2008). Aerodynamics of High-Lift Wings with Ground Effect for Racecars. doi:10.4271/2008-01-0656 (accessed on 2024-5-18).