Numerical Analysis of the Aerodynamic Contribution of the Front Wing, Rear Wing, and Diffuser on a Formula 1 Car: A Full versus Stripped Configuration Study

Anitej Dushyant Pota

Abstract

This study aims to see whether the presence of major aerodynamic components affects the drag coefficient and downforce of a Formula 1 car by comparing a full cart model against a stripped-down model without the aerodynamic components. Simulations were conducted in ANSYS Discovery 2026 R1 at inlet velocities of 50, 100, and 200 mph to accurately represent the speeds Formula 1 cars reach. The simulations reproduced the expected quadratic relationship between force and velocity to within approximately eight percent across a fourfold change in speed, confirming that the solver captured fundamental aerodynamic behavior. However, the difference in aerodynamic coefficients between the two configurations was small, ranging from roughly two to eight percent, which falls within the estimated uncertainty of the simulation method. The study therefore reports a null result: under the meshing fidelity available in Explore mode, the aerodynamic contribution of the removed components could not be resolved. This outcome is interpreted not as evidence that these components are aerodynamically unimportant, since the established literature demonstrates otherwise, but as an indication of the resolution limits of GPU-based exploratory computational fluid dynamics (CFD).

Keywords: Computational Fluid Dynamics, Formula One, aerodynamics, Drag Coefficient, Downforce, ANSYS Discovery, Computer-Aided Design.

Introduction

Background

Aerodynamic components on Formula 1 vehicles have always played a critical role in the performance and design of cars, especially in the modern era. These components affect not only the straight-line performance of the car but also its ability to corner at high speeds of over 100 mph. Components such as the front wing, rear diffuser, and rear wing channel air through and around the vehicle, effectively creating suction that pulls the car toward the ground. This study focuses specifically on the high-speed, straight-line context of Formula 1, where minimizing drag while keeping the car planted to the ground is crucial but requires tradeoffs. Because active aerodynamics is banned in Formula 1, the fixed design of these components plays a major role in how the car moves through the air. This project uses CFD (Computational Fluid Dynamics) tools to measure the drag coefficient and downforce with and without these components to determine whether their combined effect can be isolated and quantified.

Literature Review

Research on Formula 1 aerodynamics has been extensive, but several gaps remain in the literature reviewed here. Existing work documents how individual components such as front wings, rear wings, and diffusers affect drag and downforce, the two variables collected in this study. CFD has also become a standard tool for this type of research, allowing students and professionals to test configurations cost-effectively without expensive wind tunnel testing.

Regarding front wings, the research shows that angle of attack is a major factor affecting the amount of downforce generated, with the tradeoff that it also increases drag. A CFD study on front wings found that endplate design plays a significant role as well: taller endplates improve airflow around the wheel wake, reducing overall drag and improving stability (Castro and Rana).

Studies on rear wings show a similar tradeoff between downforce and drag. Bhatnagar's optimization study on front and rear wings found that the best lap times occurred when the car carried the lowest possible drag coefficient for a given downforce level, demonstrating that straight-line performance is strongly affected by rear wing configurations optimized for high downforce (Bhatnagar). Full-car CFD simulations, such as the aerodynamic analysis of a 2017 Formula 1 car using OpenFOAM, show that individual components do not act in isolation: the front wing generates vortices that directly influence the diffuser and underbody airflow. This interaction makes it difficult to isolate the contribution of a single component when studying a full car (Ravelli and Savini).

A common limitation across the existing literature is that most studies examine one component at a time, often on different base geometries and using different speeds and solver settings (Om Raj et al.). This study addresses that gap by using a single, consistent Formula 1 model and comparing a fully equipped car against a stripped baseline under identical conditions.

Objectives

Based on the literature review, the main objectives of this project are:

1. Identify whether the presence of aerodynamic components measurably affects the drag coefficient and downforce of a Formula 1 car under straight-line conditions, by comparing a fully equipped car against a stripped baseline model.

2. Determine the magnitude of any change in Cd and downforce between the two configurations across inlet velocities from 50 to 200 mph.

3. Validate whether the results align with established aerodynamic theory, specifically that aerodynamic force scales with the square of velocity.

Methodology

Modeling

A Formula 1 car model was obtained from GrabCAD, an online library of open-source, free-to-use CAD models. The model was imported into Autodesk Fusion 360 to modify the geometry before being exported as a STEP file. The STEP file was then imported into ANSYS Discovery 2026 R1 for simulation. Two configurations were tested: a full car with the front wing, rear wing, and diffuser intact, and a stripped baseline with those components removed. Both configurations were derived from the same base geometry and simulated in the same project file, with only the aerodynamic bodies suppressed between runs. This ensured that the enclosure dimensions, boundary conditions, solver fidelity, and force monitor definitions remained identical across all six simulations, so that any measured difference could be attributed to the geometry alone.

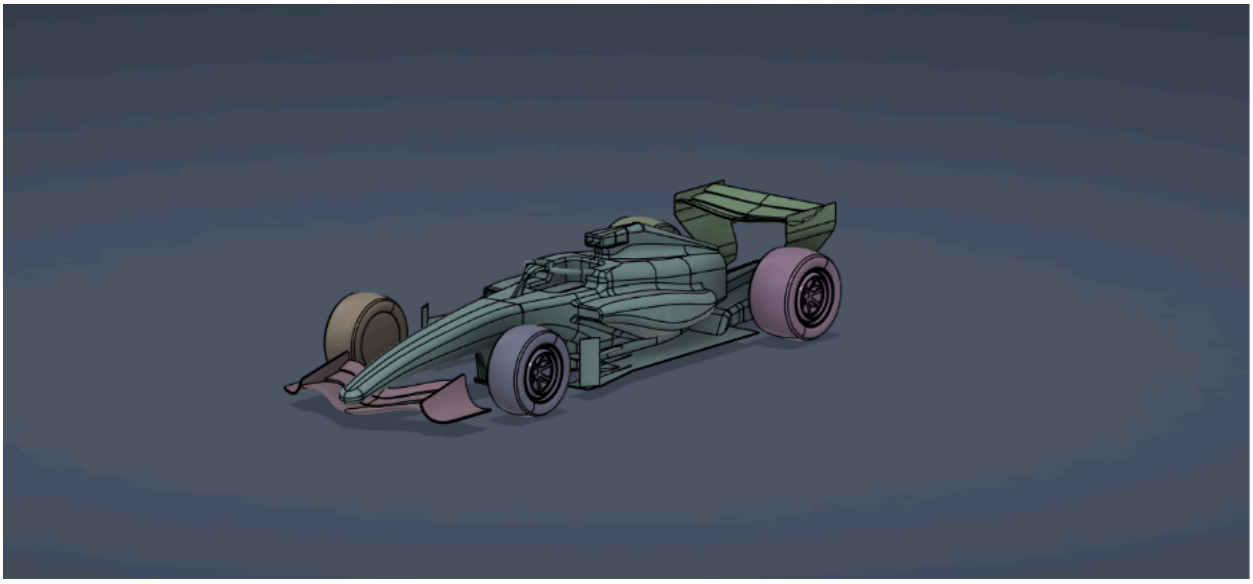


Figure 1: Full-car configuration in Autodesk Fusion 360

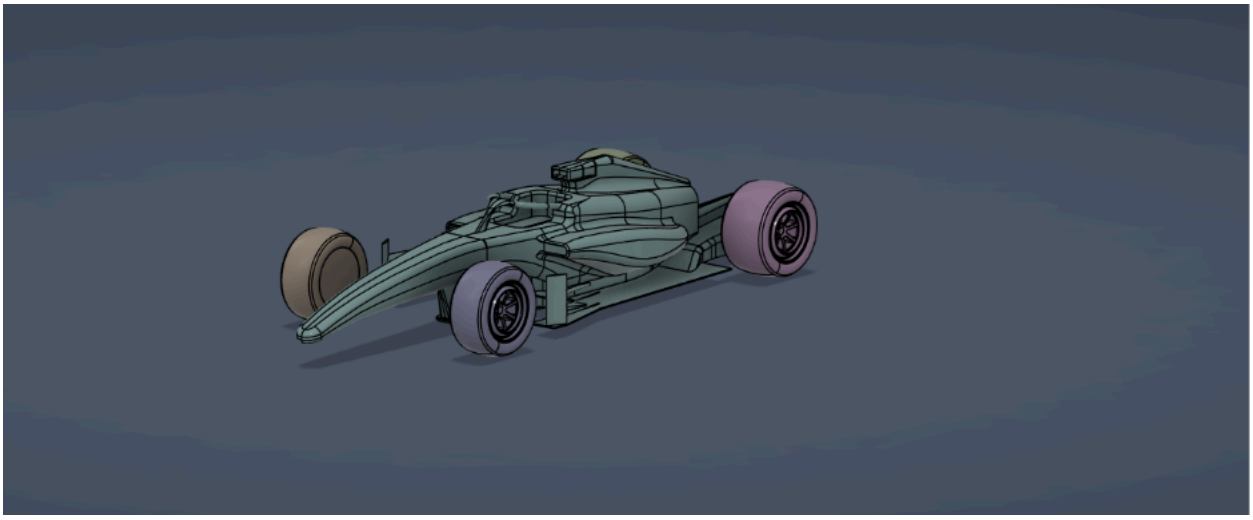


Figure 2: Stripped-down configuration with aerodynamic components removed

Key Inputs

Air was modeled as an ideal gas at standard atmospheric conditions of 22°C and 1 atmosphere. Simulations were conducted at inlet velocities of 50, 100, and 200 mph (22.4, 44.7, and 89.4 m/s) . The primary outputs recorded were drag force, downforce, drag coefficient, and downforce coefficient. These were calculated using the following equations:

$$F_d = \frac{1}{2}\rho V^2 C_d A$$

$$F_l = \frac{1}{2}\rho V2ClA$$

where Fd is drag force, Fl is the vertical force, reported here as downforce, ρ is air density (1.225 kg/m³), V is freestream velocity, C_d and C_l are the drag and downforce coefficients, and A is the frontal reference area of the vehicle. The dynamic pressure of the flow is given by $q = \frac{1}{2}\rho V^2$. A frontal reference area of A = 1.5 m² was used, which is approximately the real frontal reference area of an F1 car. Because the same reference area was applied to both configurations, the choice of A affects the absolute magnitude of the reported coefficients but not the relative comparison between configurations, which is the primary goal of this study.

Table 1: Physics and boundary-condition setup in ANSYS Discovery

Parameter	**Settings**
Solver Type	Steady-state fluid flow
Fluid	Air (Gas), ideal gas
Solid (vehicle)	Structural steel, S275N
Fluid temperature	22°C
Air density (ρ)	1.225 kg/m³

Gravity	Negative Z direction
Inlet (front face)	Velocity inlet: 22.4, 44.7, and 89.4 m/s (50, 100, 200 mph) at 22°C
Outlet (rear face)	Pressure outlet, 0 Pa gauge
Vehicle surface	Stationary wall, insulated
Remaining enclosure faces	Free slip
Enclosure floor	Flush with vehicle underside (zero clearance)
Meshing	Automatic Cartesian
Convergence criterion	Stabilization of force, velocity, and pressure monitors
Frontal reference area (A)	1.5 m²

All simulations for this experiment were done in ANSYS Discovery 2026 R1 using explore mode, which is the mode in the software that is less demanding for my GPU (Nvidia GTX 1650.) My GPU was one of the main things that held me back during the experiment. Explore mode uses automatic Cartesian meshing. A rectangular–cubic enclosure was used around the vehicle model, extending on every side of the car except the bottom, which was flushed with the underside of the vehicle model. A velocity inlet was added to one of the fronts facing sides, and a 0 Pa Pressure outlet was added directly opposite the inlet. Force monitors were scoped to the fluid-solid interface of the vehicle surface to record the total force components in the X (streamwise) and Z (vertical) directions. The vertical axis was oriented with positive Z pointing upward through the vehicle, so the negative Z values recorded by the monitors correspond to a net downward force, that is, downforce. Downforce is therefore calculated as positive.

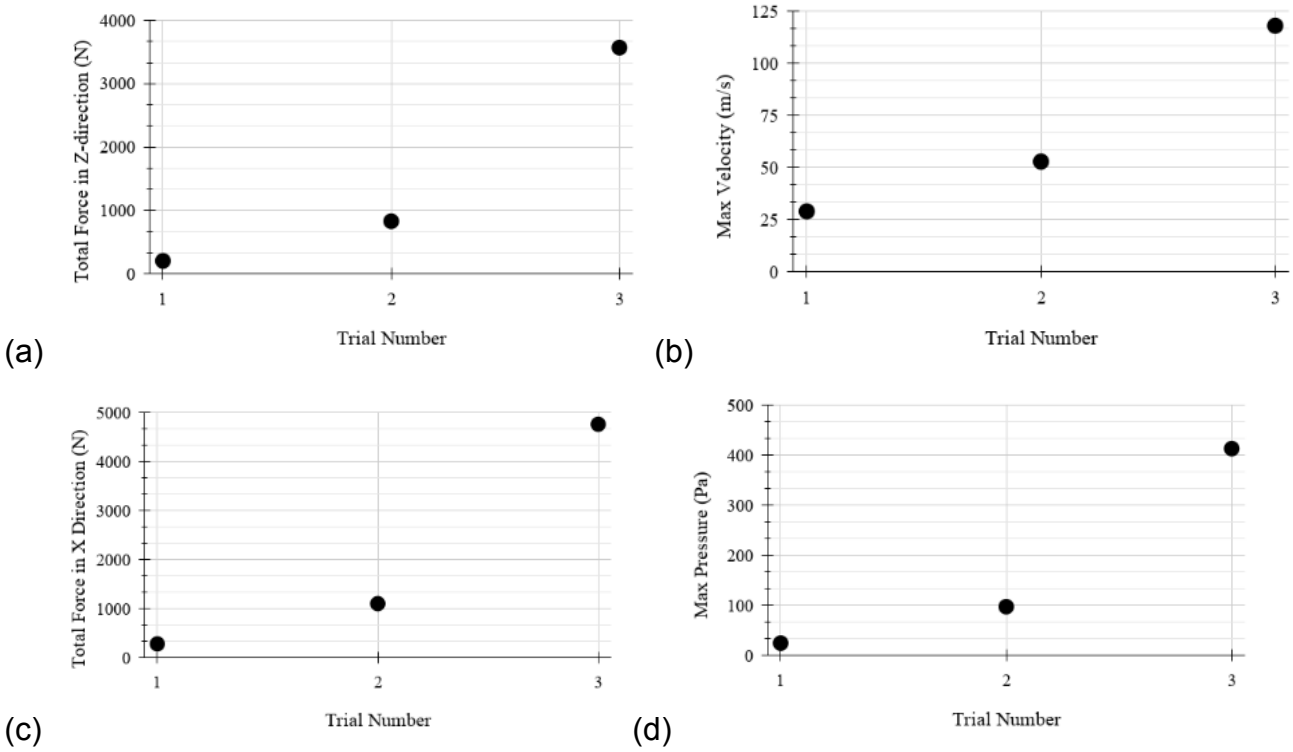


Figure 3: (a): Total Force in the Z Direction (b) Max velocity in m/s (c) Total Force in X Direction (N) (d) Max Pressure (Pa) for Trials 1, 2, and 3. Trials 1 2 and 3 correspond to 50, 100 and 200 mph inlet velocities.

Verification and Validation

Several sources of uncertainty are acknowledged in this study. First, Discovery's Explore mode uses automatic Cartesian meshing rather than a body-fitted mesh, which limits near-wall resolution and boundary layer capture, introducing approximation error particularly around thin, curved surfaces such as the front wing and diffuser. Second, the ANSYS Student license restricts Refine mode to 1,024K nodes, making high-fidelity mesh convergence studies infeasible. As a result, all results should be interpreted as approximate and suitable for relative comparison between configurations rather than as absolute validated values. To verify physical plausibility, drag force results were checked to confirm that they scale with the square of velocity, consistent with established aerodynamic theory. Qualitative flow visualization, including velocity contours and pressure distributions, was also used to confirm that the simulated flow field was physically reasonable. As discussed in Section 3, the magnitude of the difference measured between configurations must be interpreted against this uncertainty.

Results and Discussion

Simulation Results

Table 2 presents complete results for both configurations across all three test velocities. Drag force and downforce were recorded directly from the force monitors, and the corresponding coefficients were calculated using the equations given previously.

Table 2: Drag force, downforce, and corresponding aerodynamic coefficients for the full and stripped configurations at each test velocity.

Speed (mph)	V (m/s)	Configuration	Drag Force (N)	Cd	Downforce (N)	Cl
50	22.4	Full	279	0.61	202	0.44
50	22.4	Stripped	272	0.59	219	0.48
100	44.7	Full	1,100	0.60	828	0.45
100	44.7	Stripped	1,130	0.62	862	0.47
200	89.4	Full	4,750	0.65	3,570	0.49
200	89.4	Stripped	4,540	0.62	3,490	0.48

Verification of Velocity-Squared Scaling

The first validation check was whether the measured forces scale with the square of velocity, as predicted by the standard aerodynamic force equations. If this relationship holds, doubling the inlet velocity should increase the measured force by a factor of four. Table 2 summarizes the measured force ratios for each doubling of speed.

Table 3: Measured force scaling ratios for each doubling of inlet velocity, compared against the theoretical value of 4.00.

Configuration	Quantity	Ratio 50→100 mph	Ratio 100→200 mph	Theoretical
Full	Drag	3.94	4.32	4.00
Full	Downforce	4.10	4.31	4.00
Stripped	Drag	4.15	4.02	4.00
Stripped	Downforce	3.94	4.05	4.00

All eight measured ratios fall between 3.94 and 4.32, deviating from the theoretical value of 4.00 by no more than approximately eight percent. Correspondingly, the drag coefficient remains approximately constant across a fourfold change in velocity, varying between 0.60 and

0.65 for the full configuration and between 0.59 and 0.62 for the stripped configuration. This behavior is consistent with established aerodynamic theory, which predicts that aerodynamic coefficients remain approximately constant over this Reynolds number range while the forces themselves scale quadratically with velocity. This confirms that the solver captured the fundamental physics of the flow correctly, and it satisfies the third objective of this study.

Comparison Between Configurations

The central objective of this study was to determine whether removing the front wing, rear wing, and diffuser produces a measurable change in aerodynamic performance. Table 3 presents the percentage difference between the two configurations at each velocity, calculated relative to the stripped baseline.

Table 4: Percentage difference in drag and downforce for the full configuration relative to the stripped configuration. Positive values indicate a higher value for the full car.

Speed (mph)	Drag difference (%)	Downforce difference (%)
50	+2.6	−7.8
100	−2.7	−3.9
200	+4.6	+2.3

The differences between the two configurations are small and inconsistent in direction. The full car produced marginally more drag at 50 and 200 mph but marginally less at 100 mph, and produced less downforce than the stripped car at 50 and 100 mph but more at 200 mph. Across all cases, the magnitude of the difference ranges from 2.3 to 7.8 percent.

This magnitude must be interpreted against the uncertainty of the method described in Section 2.4. Explore mode's automatic Cartesian mesh has limited near-wall resolution, and the student license node limit made a mesh convergence study infeasible, so differences of only a few percent cannot be confidently separated from numerical error. In addition, the direction of the difference is not consistent across velocities, which is characteristic of numerical noise rather than a true physical effect. As a result, these results do not provide support for a difference between the two configurations. This study consequently reports a null result: under the conditions tested, the aerodynamic contribution of the removed components could not be resolved.

Interpretation of the Null Result

Although a trend can be seen in the numbers, the conclusion this paper reached highlighted another conclusion: this method of simulation could not detect a large effect of aerodynamic components. The established literature shows that the use of these components result in positive effects for Formula 1 cars. A Real Formula 1 car stripped of these components would lose most of its downforce and consequently lose performance. The finding of this study

is therefore not that aerodynamic components are unimportant. That the simulation approach used here was unable to resolve the contribution of aerodynamic components. Several factors plausibly explain this outcome. First and most significantly Explore mode uses Cartesian mesh instead of a body-fitted mesh. Wings and diffusers are highly curved surfaces whose aerodynamic function depends on resolving the pressure difference between their upper and lower surfaces and on capturing attached boundary layer flow. A coarse Cartesian grid may not place cells across these thin aerodynamic components to represent them as functioning aerodynamic surfaces rather than as generic blockage. Second diffuser performance depends on ground effect, which requires resolution of the narrow gap between the underbody and the road surface. If this gap is under-resolved, the accelerated underbody flow that generates the majority of a Formula 1 car downforce will not develop, and removing the diffuser will consequently produce little measured change. Third, the downforce magnitudes recorded were comparable to the drag magnitudes in both configurations with downforce coefficients between 0.44 and 0.49. A real Formula 1 car typically generates more downforce than drag. This discrepancy is itself evidence that the underbody and wing flow structures responsible for downforce generation were not being fully captured by the simulation.

Taken together these observations suggest that GPU-based exploratory CFD while well suited to resolving bulk flow behavior and to reproducing scaling relationships such as the velocity squared law demonstrated in Section 3.2 is not sufficient for isolating the contribution of individual thin aerodynamic components on a complex full car geometry. This is a methodological finding particularly for students and independent researchers who rely on accessible simulation tools as it indicates the point at which such tools reach the limits of their resolution.

Limitations and Future Work

In addition to the limitations of the meshing method, additional constraints also apply to this study. The frontal reference area, which is a key component in the equations previously discussed, was estimated from a typical full-sized Formula 1 car dimensions rather than measured directly from the model geometry, which minorly affects some of the results, because it doesn't affect the relative comparison between configurations, which is the goal of this study. In addition to this, only 3 velocities were tested rather than a greater number, which limits the amount of detail in any velocity dependent trend.

I believe future work in this area should use more powerful computers and software in refine mode to better capture the different aerodynamic components to make. Also, a direct measurement or a true to size cad model would improve the physical fidelity of the ground effect. A useful intermediate step would be to isolate a single component, such as the rear wing alone.

Conclusion

This study compared a full Formula 1 car CAD model against a stripped baseline model with the aerodynamic components including the front wing, rear wing, and diffuser removed. The car was simulated across inlet velocities of 50, 100, and 200mph in ANSYS Discovery. The simulations successfully represented the relationship between velocity squared and the scaling of aerodynamic force. However, the difference in aerodynamic coefficients between the two configurations was between 2.3 and 7.8 percent and was inconsistent in direction across velocities, placing it within the estimated uncertainty of the simulation method. Therefore, we have a null result according to the original research question. Given that real life applications of these components in formula one cars produce a large effect on the performance of these vehicles, the outcome in this specific experiment highlights a specific limitation of the automatic cartesian meshing within ANSYS Discovery. It can also be concluded that GPA-Based CFD simulations are fairly good for studying simple flows, but resolving complex flows around aerodynamic components on a full car geometry requires a higher fidelity meshing, and a stronger GPU, which was not present at the time of the experiment.

Works Cited

[1] Bhatnagar, Unmukt R. Formula 1 Race Car Performance Improvement by Optimization of the Aerodynamic Relationship Between the Front and Rear Wings. 2014. Pennsylvania State University, Master's thesis. https://etda.libraries.psu.edu/catalog/23699.

[2] Castro, Xabier, and Zeeshan A. Rana. "Aerodynamic and Structural Design of a 2022 Formula One Front Wing Assembly." Fluids, vol. 5, no. 4, 2020, article 237. https://doi.org/10.3390/fluids5040237.

[3] Om Raj, R. R., et al. "Design and CFD Analysis of a Rear Wing of a Motorsport Car." International Journal of Vehicle Structures and Systems, vol. 14, no. 1, 2022. https://doi.org/10.4273/ijvss.14.1.14.

[4] Ravelli, Umberto, and Marco Savini. "Aerodynamic Simulation of a 2017 F1 Car with Open-Source CFD Code." Journal of Traffic and Transportation Engineering, vol. 6, no. 4, 2018, pp. 155-63. https://www.davidpublisher.com/index.php/Home/Article/index?id=36717.html.