Clean Air Vs Dirty Air in Formula 1 Cars

by Jana Publication & Research

Submission date: 13-Sep-2025 04:43PM (UTC+0700)
Submission ID: 2690332272
File name: IJAR-53798.pdf (493.67K)
Word count: 1600
Character count: 7290

Clean Air Vs Dirty Air in Formula 1 Cars

Abstract:

Formula 1 is a famous motorsport, and the element that sets it apart from other motorsports is the forces that are generated on the cars in the dynamic environment. Teams spend millions of dollars to extract the best performance from their cars, to take corners on the track at the highest speed possible. This project report is going to talk about how a car drafting behind another car affects the drag, downforce, and pressure on a car. To get data, I will use Blender to make 3d models, and then use Ansys to do the fluid dynamics calculations.

Introduction:

In the world of Formula 1, where races are often decided by fractions of a second, aerodynamics plays a crucial role in determining vehicle performance. The aerodynamic design of an F1 car directly impacts two key forces: drag, which resists forward motion, and downforce, which increases grip and stability by pressing the car onto the track surface. Striking the right balance between minimizing drag and maximizing downforce is essential for achieving optimal lap times and ensuring vehicle control, especially during high-speed cornering and braking.

Formula 1 engineers employ a variety of aerodynamic features such as front wings, rear wings, diffusers, and underfloor tunnels, which are all shaped and positioned to control airflow. Each of these elements affects how air moves around and under the car, creating pressure zones, vortices, and flow separation. Even subtle changes in surface curvature, wing angle, or sidepod shape can lead to significant variations in aerodynamic behavior.

Computational Fluid Dynamics (CFD), as the name suggests, is a branch of fluid mechanics where computers use numerics to simulate and analyse fluids in a given condition. This allows fluid with various physical properties to be tested in various simulated scenarios, which otherwise may cost a lot to replicate in real life. This acts as a method to save money and gain information that is accurate up to a certain degree. CFD allows one to create a frame/design, alter physical properties of fluid and the object, and gain numerical values for various aspects such as force, pressure, vortices, continuity, and more.

Dirty air significantly changes the aerodynamics of a car. Dirty air is the air that has gone through another car and is not in a steady state, but a turbulent one. Since turbulence is not completely predictable, dirty air makes a car behave unpredictably. Leaving the drivers to rely on their experience and skills. Understanding how dirty air exactly affects a car will be very beneficial to become faster and compete in combats in F1.

Current issues in the CFD that are limiting the research into this field are the unpredictability of turbulence. Till now, we haven't been able to find a definitive model to perfectly simulate turbulence. Several relations have been found between different features of a car and turbulence, but none of them reach a general formula. This paper will also discuss some existing models of calculating turbulence.

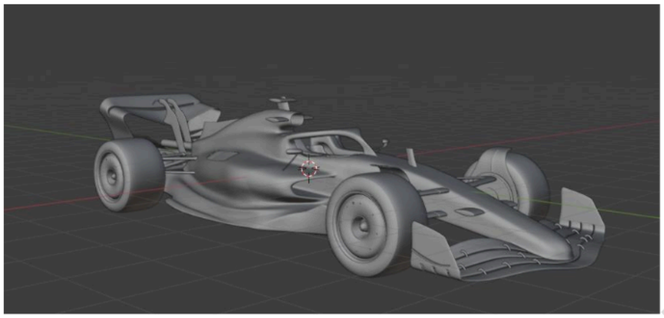
This paper aims to analyze how a car drafting behind another car affects the drag and downforce experienced by a car. Secondly, analyse how dirty air affects cars. All of this will be solved using principles of fluid dynamics and computational simulations. By understanding these interactions, the research seeks to highlight effective aerodynamic strategies that enhance race performance.

F1 Car Image:

Methodology:

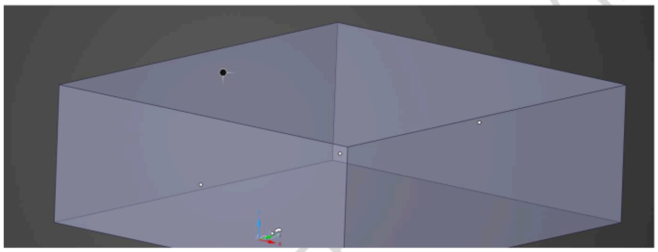
To gather information about a standard F1 car, one will have to make an F1 car in Blender that is roughly similar to the ones in real life. This will allow gathering information on current standards. I have taken a premade F1 car model in Blender. Then, performance information will be gathered using Ansys, which is a CFD. Then I will place another car that is the same in front of the car. Using the data collected, I can then compare the data sets. This will help gain insights into the benefits and drawbacks of drafting behind a car and what causes it. Most of the comparison will be done on a spreadsheet as it's a very easy and reliable tool. This way, we will conclude how drafting affects the aerodynamics of a car.

Fig 1



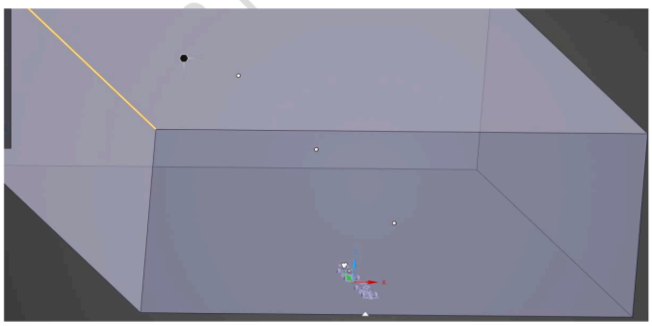
The F1 car model used to simulate

Fig 2



The clean air setup is to simulate

Fig 3



The dirty air setup is to simulate

Results:

The results from the CFD can be accessed from this link (graph given under the Data header). There is an anomaly in the readings, which is not easy to miss, that does not represent how fluid behaves in real life. The dirty air drag is not only greater than all other datasets, but also greater than the drag in clean air. In real life, the opposite is seen, when cars are behind each other, less drag is created due to wakes from the leading car.

However, the other variables, pressure and downforce, align with real-life behaviour. Both the downforce and pressure faced on a car are reduced when it is in dirty air than in clean air. This is because the car in front displaces the air particles in front and reduces the volume and velocity of particles that the car interacts with. Since lower pressure means lower force, there will be lower downforce experienced as well.

The reason for the anomaly in drag might lie in the 3D model for the simulation. Specifically, the car might have an unintentional opening that is directing air inside the car and leading it outside through some unknown opening. F1 cars in real life do channel air via internal ducts to boost performance; however, the model I used didn't have detailed insides, which can lead to inaccurate fluid flow and, thus, misleading data.

The equations Ansys CFD uses to calculate data are the Navier-Stokes Equations: Continuity Equation (Mass Conservation)-

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \vec{u} = 0$$

The Motion Equation (Momentum Conservation)-

$$\rho \frac{D\vec{u}}{Dt} = -\nabla p - \nabla \cdot \tau + \rho \vec{g}$$

Energy Conservation Equation

$$\rho c_\mu \frac{\partial T}{\partial t} + \rho c_\mu U_i \frac{\partial T}{\partial x_i} = -P \frac{\partial U_i}{\partial x_i} + \lambda \frac{\partial^2 T}{\partial x_i^2} - \tau_{ij} \frac{\partial U_j}{\partial x_i}$$

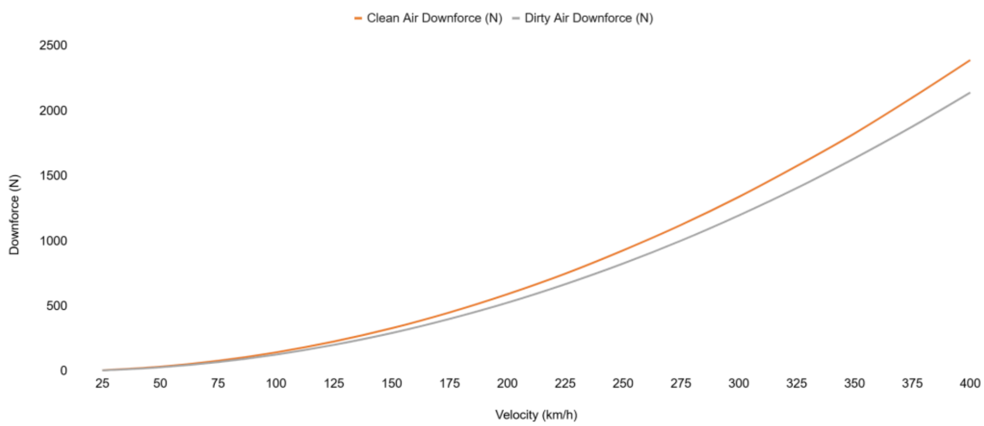
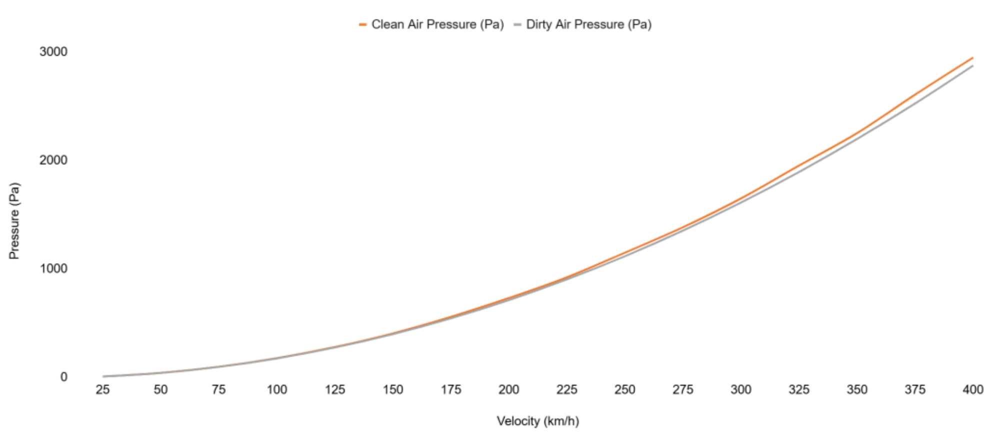
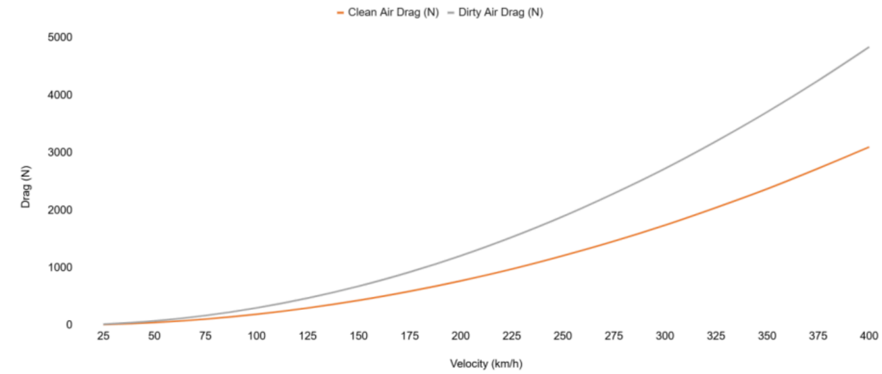

Clean Air Downforce (N)
Dirty Air Downforce (N)
Downforce (N)
2500
2000
1500
1000
500
0
25
50
75
100
125
150
175
200
225
250
275
300
325
350
375
400
Velocity (km/h)

Fig 4


Clean Air Pressure (Pa)
Dirty Air Pressure (Pa)
Pressure (Pa)
3000
2000
1000
0
25
50
75
100
125
150
175
200
225
250
275
300
325
350
375
400
Velocity (km/h)



Conclusion:

Other than the wrong data on the drag, the data can help us lead to quite a few conclusions. Firstly, drafting leads to greater acceleration. This can be proved by two reasons: the drag is less, so the net force will be greater towards the front, and the decreased downforce results in smaller rolling friction, as weight is directly proportional to rolling friction. This can help gain an acceleration advantage when trying to overtake another car.

The decreased downforce can be explained by pressure from incoming air particles. When in clean air, the F1 car wings are hit by the air particles in their way. Due to Newton's third law, every action has an equal and opposite reaction; some force is applied to the car. Since the car's front and rear wings are angled and produce some amount of downforce and drag, using vectorization of force, it can be calculated. On the other hand, when a car is behind another car, it faces fewer air particles (as they are flowing around it due to another car being in front), which means the resultant force of the air particles is less. This would reduce the magnitude of the downforce faced by the car. It will also increase top speed as the drag has also been reduced for the same reason.

However, there are a few drawbacks to this phenomenon. Firstly, the decreased downforce means the car is pushing less on the ground, essentially reducing its tire grip. This means at a corner, the leading car can take the turn at a greater speed than a car that is drafting. Another drawback is turbulence, which is impossible to predict and can reduce the confidence of the driver in the car. In racing, the driver needs to be able to understand the car, but under turbulence, it gets hard, as there is no predictive pattern.

For a strategy that can be used in racing, it is best to draft when on a straight part of the track, where the next corner is far away; this will help the driver maximise the acceleration and outpace the lead car. The driver does not need to predict the car much on a straight as the car tends to go in a straight line, and there is not much precision needed anyway as there is no turn. However, if a turn is coming up, it is good to back off from the tail of a car as much as possible without giving up too much time at the corner. This is because the car needs more braking power, which reduces under-drafting due to lower drag, and the reduced downforce. Even though at first it feels wrong to back off, in fact ,the grip gained from the action can help the car get a better exit on the corner, so that it can draft and overtake before the next corner comes up.

Clean Air Vs Dirty Air in Formula 1 Cars

ORIGINALITY REPORT

2%	1%	0%	2%
SIMILARITY INDEX	INTERNET SOURCES	PUBLICATIONS	STUDENT PAPERS

PRIMARY SOURCES

1	www.coursehero.com Internet Source	1%
2	Submitted to Associatie K.U.Leuven Student Paper	1%

Exclude quotes	On	Exclude matches	Off
Exclude bibliography	On		