7-18-2023

Quantifying the Impact of the 2022 Formula One Technical Regulations on Wake Turbulence: A Numerical Analysis

Luis A. Méndez

Quantifying the Impact of the 2022 Formula One Technical Regulations on Wake Turbulence: A Numerical Analysis

Honors Thesis

Luis A. Méndez

Department: Mechanical and Aerospace Engineering

Advisor: Markus P. Rumpfkeil, Ph.D.

July 2023

Quantifying the Impact of the 2022 Formula One Technical Regulations on Wake Turbulence: A Numerical Analysis

Honors Thesis

Luis A. Méndez

Department: Mechanical and Aerospace Engineering

Advisor: Markus P. Rumpfkeil, Ph.D.

July 2023

Abstract

For the 2022 Formula One (F1) season, the Federation Internationale de l'Automobile (FIA) introduced a new set of technical regulations that reduce the complexity of the aerodynamic devices such as the spoilers, often called wings. The objective of this regulation change is to reduce the amount of turbulence produced, which should allow the cars to trail behind one another closer and make for easier overtaking, thereby increasing the competitiveness of the sport. The present study evaluates and quantifies the aerodynamic performance of a 2022 F1 car by using computational fluid dynamic (CFD) analyses. Both a study of a 2022 specification rear wing and a 2021 specification rear wing are assessed to determine how the new technical regulations affect the turbulence in the wake of the car. The study is performed by taking cut planes in the fluid domain downstream of the F1 rear wing models and integrating turbulent kinetic energy across the planes to quantify the turbulence in the wake. With this analysis, a comparison between the 2022 and 2021 specification rear wings can be performed to determine the magnitude of impact the new technical regulations produce. From this, a conclusion could be made regarding the effectiveness of the 2022 F1 technical regulations, and whether the regulation change was justified.

University *of* Dayton

Table of Contents

1 | Introduction

For years, the FIA Formula One World Championship (F1) has struggled to produce competitive racing. While there are many factors that contributed to this lack of competition, including an inequality of team budget and resources, a leading cause of the lack of competition on track was the amount of turbulence produced in the wake of the car. As technology improves, the complexity of the aerodynamic devices on F1 cars has dramatically increased, however, as a result, the turbulence has also increased. F1 cars are designed to operate in laminar flow, so when trailing behind in the turbulence of another car, efficiency is compromised, limiting the capabilities of the trailing car and making overtaking difficult.

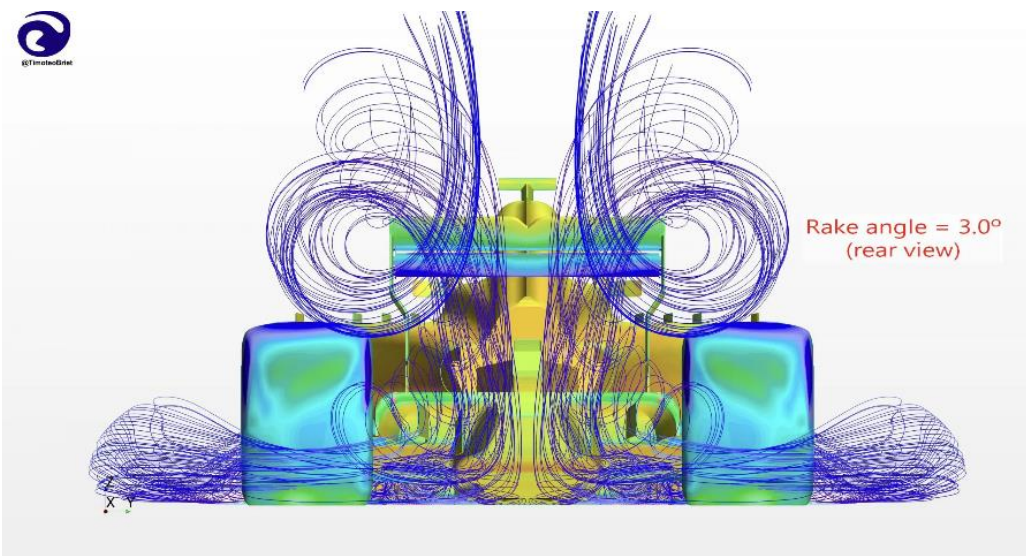


Figure 1: View of the rear of an F1 car depicting the turbulent vortices in the wake. Extracted from Analisi Tecnica [1].

After investigations performed by the Federation Internationale de l'Automobile (FIA) and F1 teams, a new set of technical regulations was agreed upon with the objective of improving the competitiveness of the sport. As a part of these changes, the regulations now restrict the complexity of the front and rear wings, which narrows the outwash produced. However, to make up for the lost downforce from the wings, the new technical regulations do allow for venturi tunnels to improve the efficiency of downforce generated by the low-pressure zone underneath the car [2]. It is alleged that the downforce generated by the underbody of the car, often referred to as ground effects, is less sensitive to turbulence. Combined, these regulation changes allow F1 cars to follow closer to one another and provide more overtaking opportunities.

Computational Fluid Dynamics (CFD) is a numerical simulation tool that enables the study of fluid flow and heat transfer phenomena in various engineering applications. It is widely used in industry and academia as a cost-effective and efficient approach to analyze complex fluid dynamics problems. CFD utilizes mathematical models and algorithms to solve the governing equations of fluid motion, including the conservation of mass, momentum, and energy. CFD has been a crucial tool in the development of F1 cars since the 1990s [3]. By using CFD simulations, engineers can analyze and optimize the aerodynamic design of the car, including the shape of the bodywork and the placement of wings and other aerodynamic components. CFD simulations also enable engineers to study the interaction between the car and the track, such as the effects of airflow around corners and the impact of crosswinds on the stability of the car. In F1, CFD has become an essential part of the design process, allowing teams to develop and test new ideas quickly and efficiently in a virtual environment before implementing them on the physical car.

Previous studies have demonstrated the feasibility of using CFD to evaluate the turbulence in the wake of an F1 car. These investigations, including the works of Ravelli and Savini [4,5], have provided valuable insights into understanding the vorticity behaviors and flow characteristics of the cars. Reference data from these studies are often considered for comparison in the assessment of numerical results under free stream flows.

The works of Newbown et al. [6] and Perry et al. [7] have also inspired researchers in numerically evaluating F1 cars under wake flows. Their methodologies, particularly the variation of distances between cars, serves to capture the effect of turbulence on a trailing car. However, as these studies use geometry based on previous technical regulations, studying the current regulations could provide a comprehensive understanding of how current F1 cars are affected by the wake flow.

The main contributions and goals of the current study are to evaluate, study, and quantify the turbulence in the wake of a geometric representation of a 2022 and 2021 F1 rear wing in free stream conditions with the purpose of documenting the differences between the wake flows and test whether the 2022 technical regulations fulfill their intended purpose. Additionally, an original approach to quantifying turbulence in the wake of the rear wing is given so as to provide a direct comparison between the two rear wing specifications.

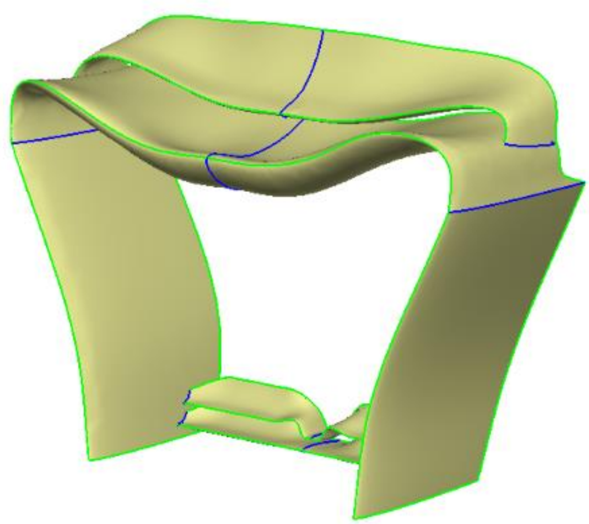
The current study can be considered contemporary, as few sources have published conclusions regarding the 2022 F1 technical regulations. The originality of the current study lies in the ability to provide additional insight and judgment on the technical regulation changes introduced for 2022 with the objective of contributing to future improvements to the sport.

2 | Methodology

2.1 Geometry

The CAD geometry for the current study was generated using The Engineering Sketch Pad (ESP), a solid modeling software developed at the Massachusetts Institute of Technology (MIT). ESP was chosen because it is optimized to develop aerospace geometry. To define the dimensions and key features for the rear wing geometry, the 2022 and 2021 technical regulations [8,9] along with supporting documents such as schematics by Giorgio Piola [10] and reference photos [11,12] were used. In Figure 2, the geometric representation of the 2022 F1 rear wing is seen alongside a photo of an actual 2022 F1 rear wing. The geometry developed for the study is not an exact replica but does capture the primary features and dimensions as outlined in the technical regulations. What is particularly notable about the 2022 rear wing is the distinctive curve as the main wing elements blend into the sides.

(a)



(b)

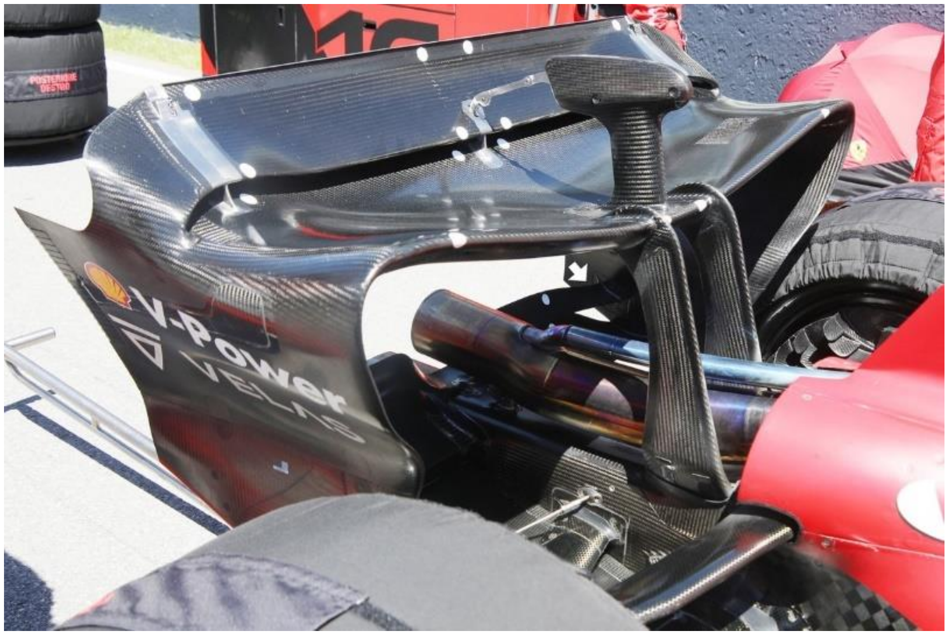
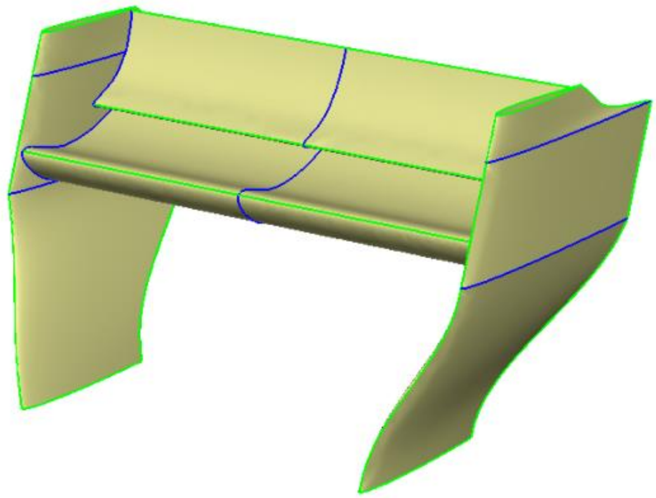


Figure 2: Comparison between the (a) geometric representation of the 2022 F1 rear wing developed in ESP and (b) 2022 F1 rear wing of the Ferrari F1-75. Extracted from Scuderia Fans [11].

This differs dramatically from the 2021 rear wing as seen in Figure 3 where the wing elements attach directly to the end plates at a 90-degree angle. This change in the technical regulations was made specifically to reduce the turbulence in the wake of the car. The stark transition from wing to endplate is notorious for generating vortices as air rolls up near the end plate as it flows over the wing. By seamlessly blending the wing elements into the end plates, the disruptions of the airflow are minimized, resulting in less turbulence in the wake of the car.

(a)



(b)

Figure 3: Comparison between the (a) geometric representation of the 2021 F1 rear wing developed in ESP and (b) 2021 F1 rear wing of the Mercedes W12. Extracted from PlanetF1 [12].

The simplifications that were required when developing the geometry of the 2021 rear wing should also be noted. The endplates of the actual rear wing split into two different sections about halfway down and include serrations at the bottom. The decision to exclude these features in the geometric representation of the 2021 rear wing was made because the computational power and time required were not available during the current study.

2.2 Simulation Setup

To define the dimensions of the fluid domain, previous works were consulted. While several domain sizes are used by authors, a cubical domain is used by all which indicates that is the most suitable for this study. Ashton et al. [13] utilized a cubical domain with dimensions of 8H height, 14H width, 13H upstream, 19H downstream, and 14H cross-stream, where H represents the height of the car. Heft et al. [14] determined the upstream (2L) and downstream (7L) sizes of the domain based on the length of the car (L). Arrondeau et al. [15] focused on a 2021 F1 front wing study and used a fluid domain with dimensions of 5H height, 3H upstream, and 10H downstream. In a more conservative approach, Simmonds et al. [16] employed a domain with dimensions of 10L height, 20L width, 12L upstream, and 15L downstream. Based upon this, the current study utilizes a fluid domain with the dimension of 3.7D height, 3.7D width, and 5.5D height, where D is the depth of the rear wing. These dimensions provide a wide space around the wing which limits the effect of any numerical instabilities and fluid recirculation caused by the boundaries of the volume. Additionally, the large downstream volume allows for numerous cut planes at various distances behind the rear wing representations to analyze the generation and dissipation of the turbulent wake.

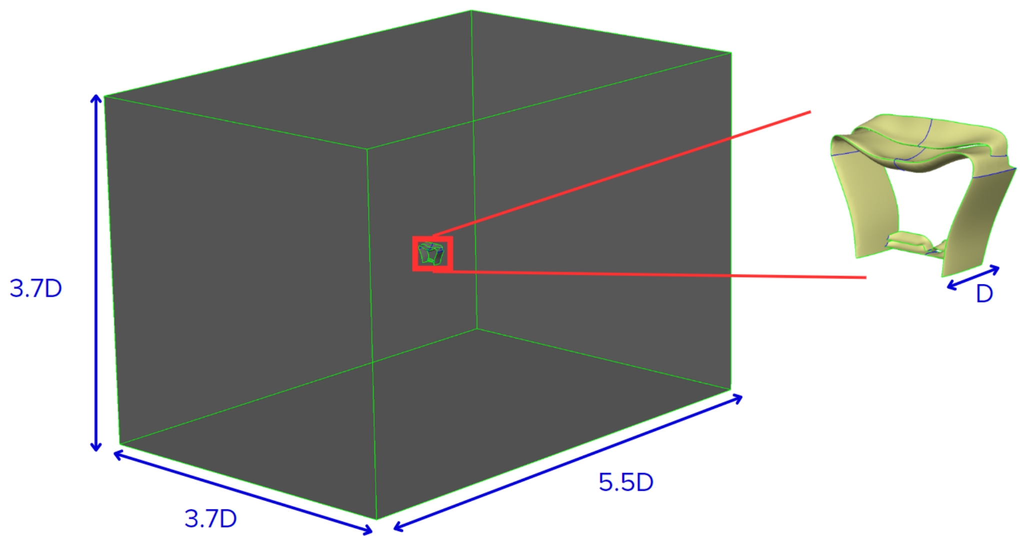


Figure 4: Depiction of the fluid domain used for the 2022 F1 rear wing model.

Refinement enclosures, depicted in Figure 5a, are specified around the geometry of the rear wing to capture in detail the effects of the wake. The mesh applied to the CAD model as seen in Figure 5b, is a hybrid mesh because it offers a high accuracy to cost ratio. The mesh is composed of a structured boundary layer and uses unstructured methods in other regions of the domain.

The fluid dynamic analysis is performed in FUN3D, a CFD software developed by NASA which is well suited to study external fluid flow. FUN3D uses a finite-volume method to solve partial differential equations governing fluid dynamic problems. The governing equations employed are the Reynold-Averaged Navier-Stokes (RANS) equations. RANS provides a lower computational cost than other models such as Large Eddy Simulations (LES). The turbulence model to be used to carry out the study is the k-ω SST model. This is a widely used turbulence model in engineering applications and motorsport aerodynamic simulations because it provides a balance between accuracy and time [17].

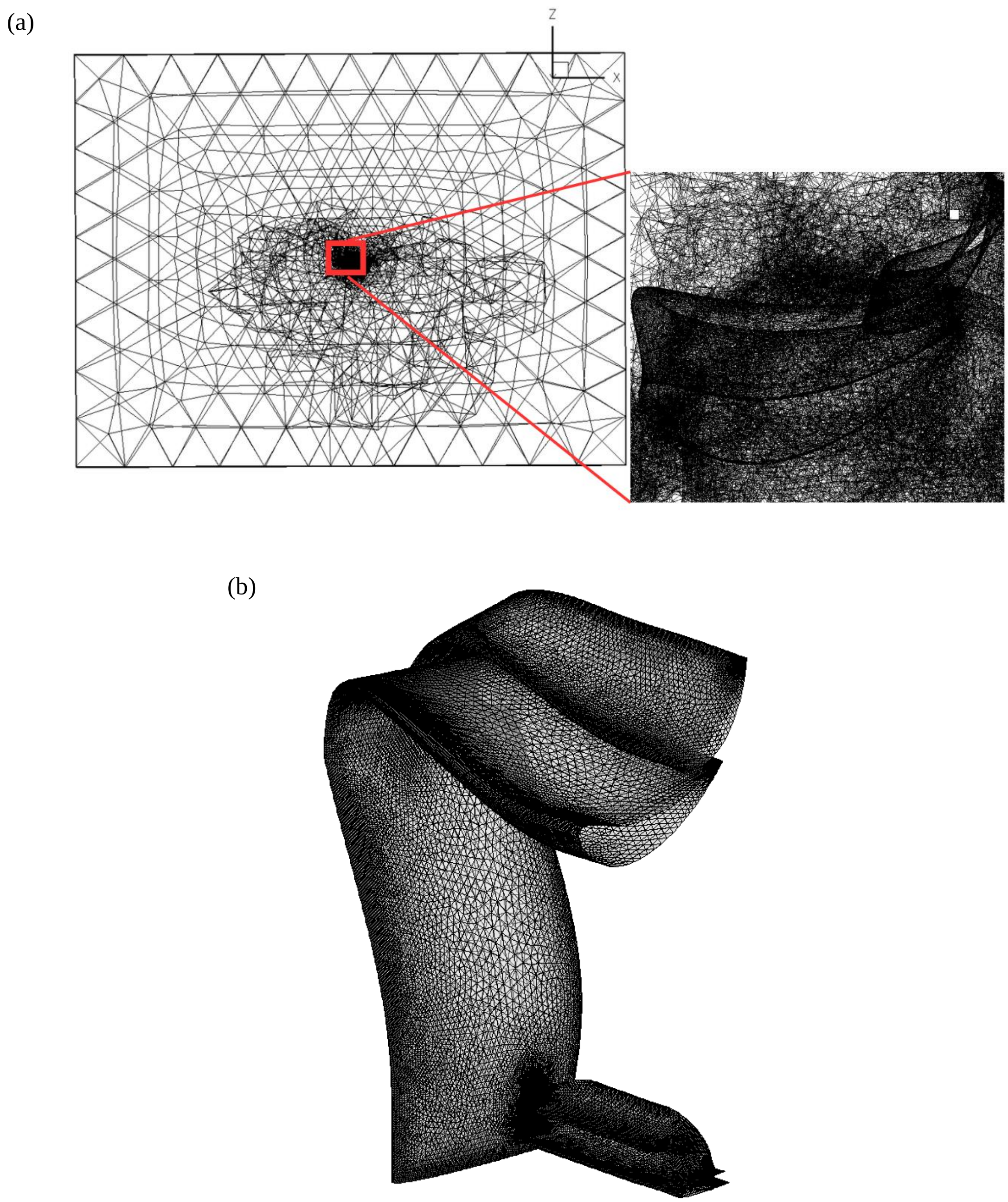


Figure 5: Meshes: (a) volume mesh of the fluid domain for the 2022 F1 rear wing with a close-up of the refinement enclosures around the rear wing; (b) surface mesh of the 2022 F1 rear wing.

The boundary conditions of the wing are a crucial aspect of setting up the CFD model to produce accurate and meaningful results. For the study of the rear wing, it is assumed that the wing is moving through the air away from any walls. For the inlet conditions, fluid conditions are assumed to be air at sea level. A mean velocity of 0.2 mach was set, which is approximately the average speed of an F1 car during a race [18]. For the outlet condition, a zero-pressure gradient was selected to allow for recirculation. Additionally, a symmetry boundary condition is selected along the plane Y=0 for the purpose of halving the size of the mesh and reducing the computational cost of the simulation.

Table 1: Summary of fluid and flow conditions.

Variable	**Value**
Free stream velocity (U_∞)	68.6 m/s
Fluid density (ρ)	1.225 kg/m^3
Fluid Temperature (T)	287.15 K
Fluid pressure (p)	101000 Pa
Dynamic viscosity (μ)	$1.81 \times 10^5\ kg\backslash m \cdot s$
Reynolds number (Re)	96300

2.3 Simulation Performance

The CFD simulations were performed on a desktop computer equipped with 8 cores and 32 GB of RAM memory. Post-processing was primarily performed on the desktop computer and supported by a laptop with four cores and 16 GB of RAM.

The model was considered converged once the residuals achieved steady state. As seen in the convergence history of the model in Figure 6, all the key quantities exhibit convergence, ultimately reaching a steady state. However, the magnitude of convergence is not ideal, with the residuals decreasing only a few orders of magnitude before achieving steady state. A decrease of at least four orders would be best. The lower convergence level suggests an insufficient mesh resolution. Despite investing considerable time in improving the convergence history, it was eventually determined that due to time constraints, a convergence of a few orders was sufficient for the scope of the project.

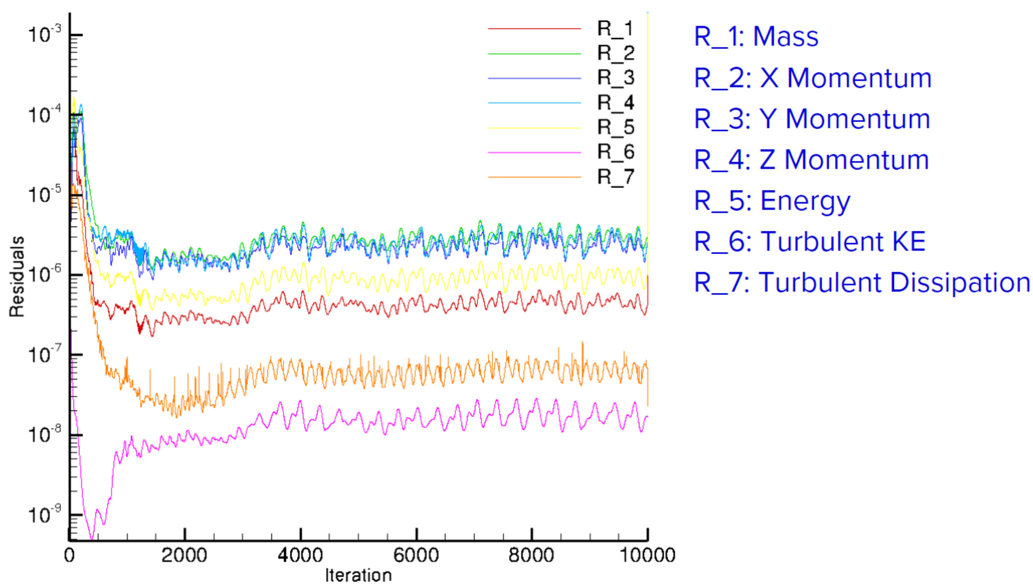


Figure 6: Convergence history of the F1 2022 rear wing model.

3 | Results and Discussion

3.1 Validation and Verification

When completing any analysis using CFD, it is important to ensure the accuracy of the results and to quantify the uncertainty of the simulation. To test the accuracy, validation and verification checks are completed. The primary concern regarding validating results is whether the correct equations were used and assumptions were made. Typically to validate the results of the CFD simulation, a comparison to real-world testing, often in the form of wind tunnel testing, is made; however, producing a wind tunnel model would be resource intensive. Instead, a comparison was made to existing literature studying the aerodynamics of 2022 F1 cars. However, finding a comparable solution proved more challenging than initially anticipated, as F1 teams do not readily share their data. Extensive internet research was conducted to locate similar CFD analyses of a 2022 F1 car, which yielded limited results. One relevant study was discovered on the F1 Technical online forum posted by the user Latios [19], enabling a comparison of the lift values between the rear wing of the known solution and the obtained solution. Lift was calculated using the following formula:

$$L = C_L \left(\frac{\rho V^2}{2}\right) A$$

The lift for the solution was calculated by first approximating the combined surface area of lift generating surfaces. Each lift generating surface was divided into rectangular and trapezoidal areas. The resulting areas were summed to calculate the total surface area of the 2022 F1 rear wing representation. The inlet conditions were used for the fluid density and velocity, and the lift coefficient output from the final iteration of the model was -1.65. Using these values yielded a lift force of -2374 N. In the CFD analysis posted by the user Latios, the lift generated by the rear wing is -2321 N. This reveals a promisingly small error of 2.25%, indicating the model used in the current study is a relatively accurate representation of a 2022 F1 rear wing.

Table 2: Drag and lift data for a full F1 2022 car CFD model [19].

Full Model @ 60 m/s	**Drag (N)**	**Lift (N)**
Front wing	279	-3137
Front body nose	72	-310
Front tire	416	329
Front wheel wing	13	60
Chassis	44	123
Floor	413	-4794
Rear tire	765	197
Rear wheel wing	82	-257
Rear profiles	553	-1693
Rear beam	328	-628
Rear wing	881	-2321
Total	2965	-10110

Verification demonstrates whether the equations were solved correctly. To verify the result of the study, a grid convergence study (GCS) was completed. By completing two additional runs of the simulation using a finer and coarser mesh, a graph can be generated comparing the desired parameter and $N^{-\frac{p}{d}}$, where N is the number of nodes, p is the order of accuracy of the CFD solver, and d is the number of spatial dimensions. If the results of all 3 meshes fall on a straight line, then the model operates in the asymptotic range of convergence and the results are grid independent, wherein the results would converge to a common value given an infinitely fine mesh. To change the mesh resolution, the grid length was adjusted to 0.5 for the fine mesh and 2.0 for the coarse mesh. The lift data from all three solutions were plotted against the grid convergence index as seen in Figure 7. The nearly linear slope across all solutions provides evidence that the mesh quality is likely adequate for the analysis conducted.

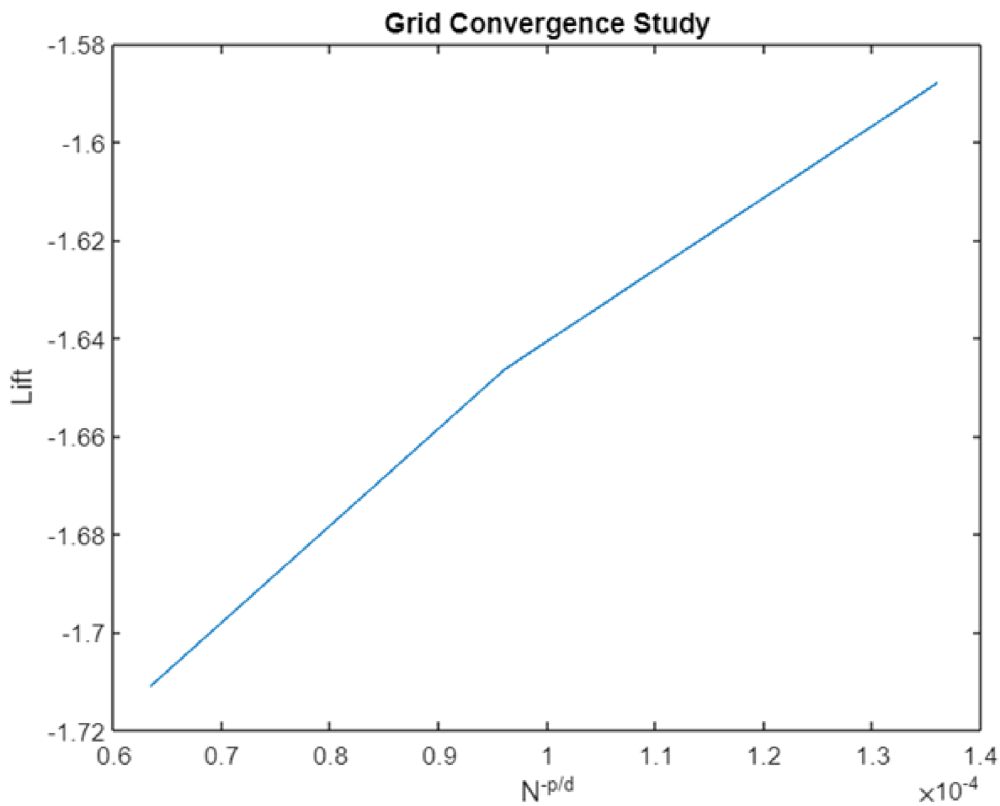


Figure 7: Grid convergence study of the lift force of the F1 2022 rear wing model.

3.2 Pressure Distribution

The pressure distributions as shown in Figure 8 provide a visualization of how lift, or in this case downforce since the lift force is negative, is generated at the middle of the wing. As air passes over the inverted wings, a region of high-pressure forms above the wing, while a region of low-pressure forms below it. This pressure difference generates a net downward force, thereby improving the grip of the car. Additionally, a noteworthy observation is the presence of a region of low pressure behind the 2021 wing, likely indicating the turbulent wake. The existence of this region solely behind the 2021 wing suggests that the regulation changes are effective to some extent, as they reduce the area of turbulence in the wake.

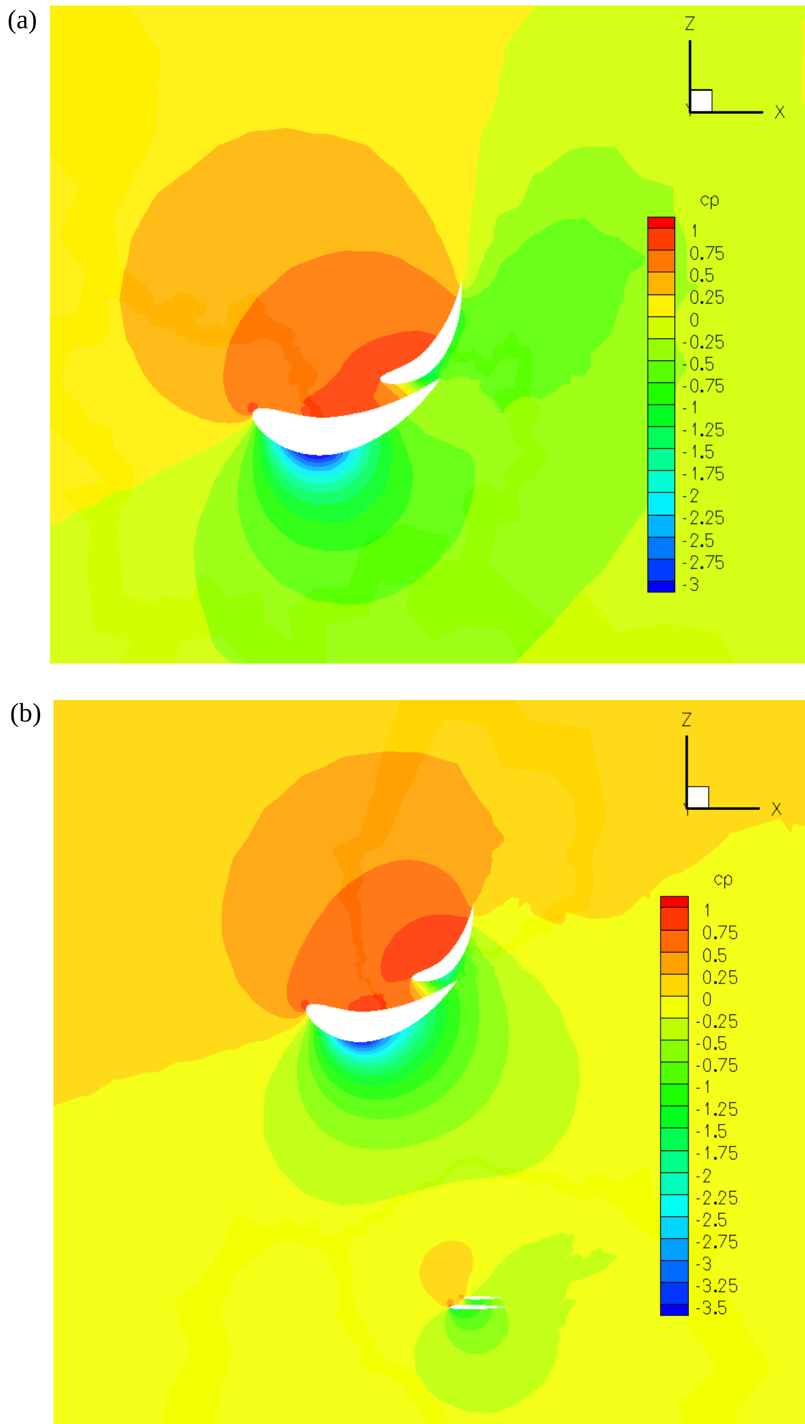
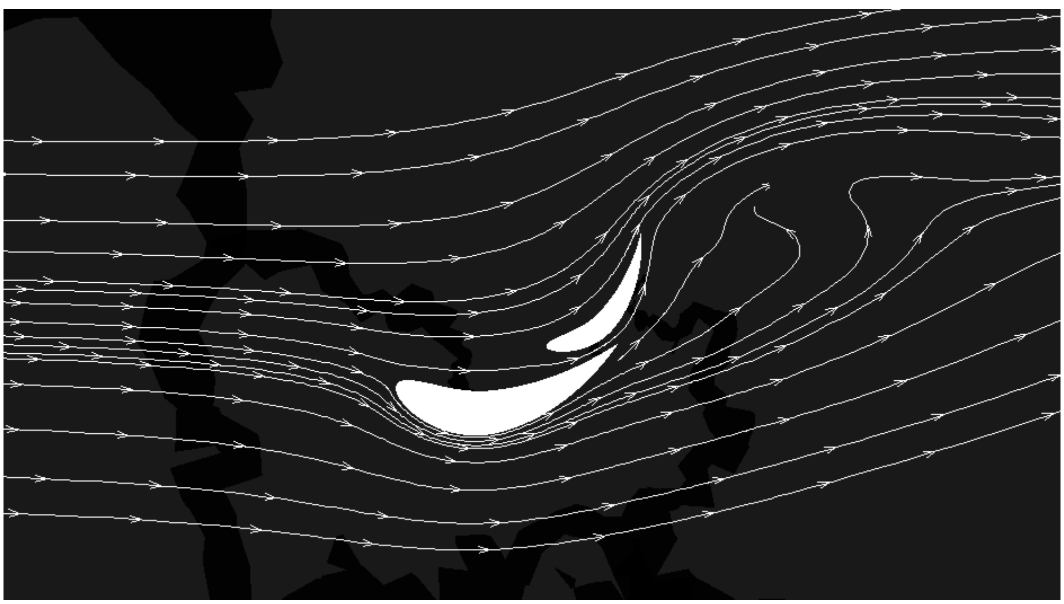


Figure 8: Cross-section of the pressure distribution at the center of the (a) 2021 F1 rear wing and (b) 2022 F1 rear wing.

3.3 Streamline Plots

Streamline plots, as seen in Figure 9, were employed to provide further evidence of the turbulent wake generated by the rear wings. These plots clearly depict the detachment of air from the wing and the subsequent rotation. The air detaches sooner on the 2021 rear wing when compared to the 2022 rear wing. The rotation observed behind the 2021 F1 rear wing in Figure 9a is the turbulence visualized. This unsteady flow will continue to permeate behind the rear wing until all the turbulent kinetic energy has dissipated. The region of air rotation behind the 2021 rear wing also aligns with the region of low pressure that was noticed in Figure 8a further reinforcing that the rotation observed is turbulence.

(a)



(b)

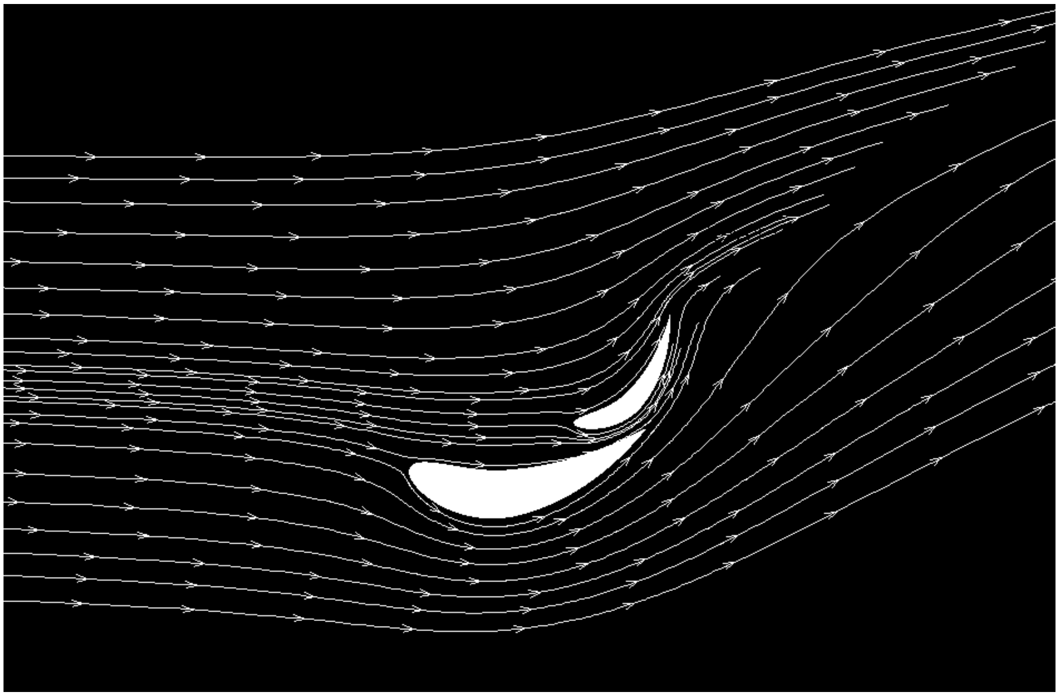


Figure 9: Cross-section of the streamline at the center of the (a) 2021 F1 rear wing and (b) 2022 F1 rear wing.

The slipstream effect can also be observed in the streamline plots. The air behind the rear wings is projected upward, effectively creating a hole in the air behind the car. This reduces the air resistance on the trailing car, allowing the car to gain extra speed down the straights. Thus, the streamline plots encapsulate the dichotomy of F1 rear wings. The wings produce turbulent air which decreases the effectiveness of the aerodynamic components vital for generating the downforce needed to corner at high speeds, but also creates a low-pressure zone behind the car which can be used to effectively increase the speed of the car down the straights [20].

3.4 Wake Cut Planes

In the cut planes seen in Figure 10, the observations align with the previous observations. Turbulence is seen across the span of the 2021 wing, while the 2022 wing concentrates the turbulence at the edge. However, due to the inclusion of a lower beam wing in the 2022 configuration, turbulence was also generated in that region. Notably, the beam wing of the 2022 configuration appears to generate more turbulence than the upper profile wing.

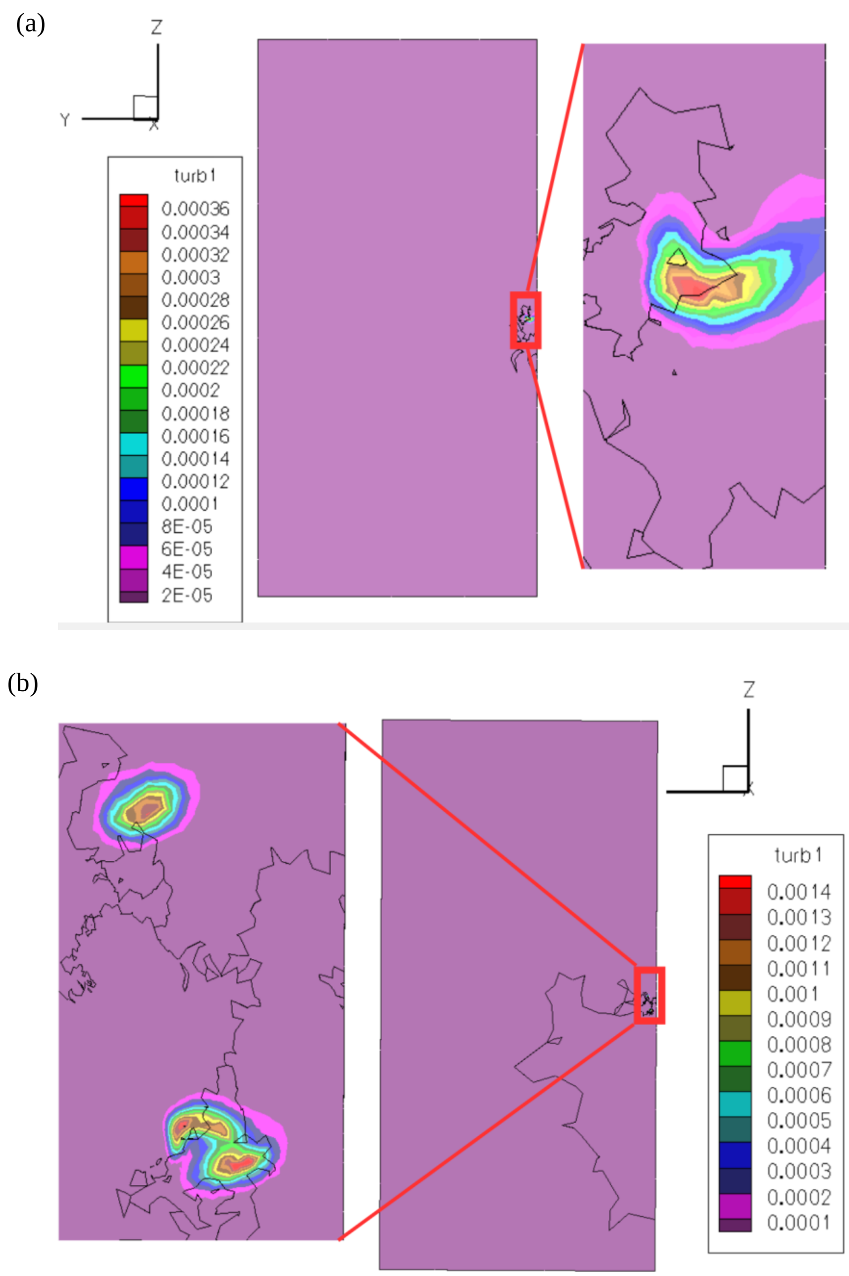
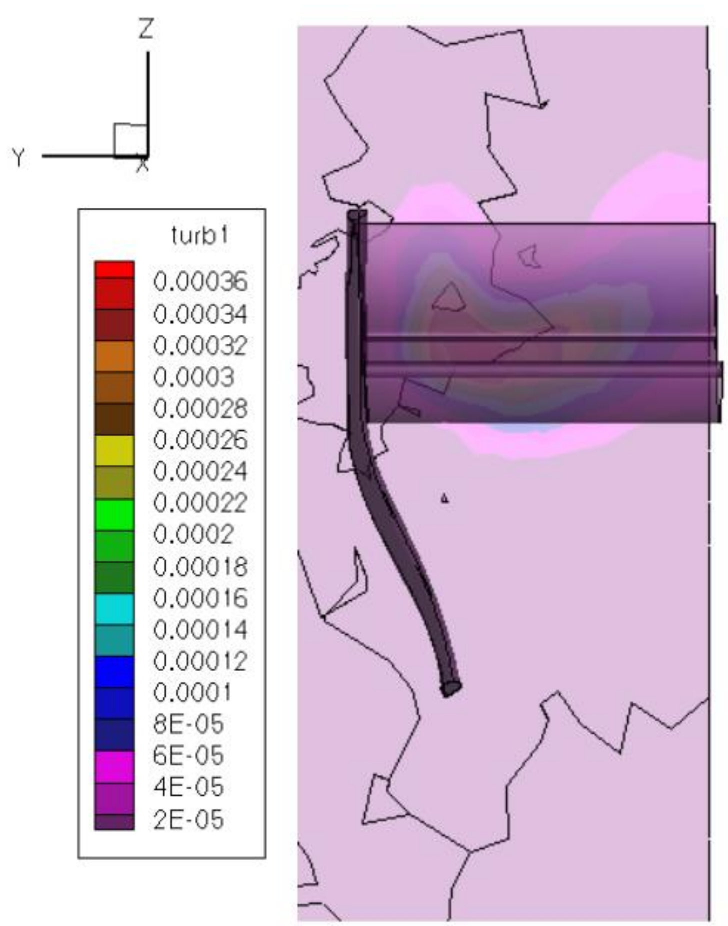


Figure 10: Wake cut planes 0.1 meters from the trailing edge of the rear wing with turbulent kinetic energy data: (a) 2021 F1 rear wing; (b) 2022 F1 rear wing.

By overlaying the wing in front of the wake cut planes in Figure 11, a clearer visualization of turbulence generation across the wing is obtained, further supporting the previous observations. It is evident that the 2022 beam wing might generate more turbulence than the profile wing due to their attachment method, potentially resulting in a sharper transition and increased turbulence. However, the turbulence generated by the beam wing is by design. The 2022 rear wing design funnels the turbulent air to the center of the car. When combined with the underbody of the car and the diffuser, the turbulence is lifted. The expansion of the air from the diffuser effectively pushes the turbulence up and over the trailing car, reducing the turbulence observed in the wake of the car [21].

(a)



(b)

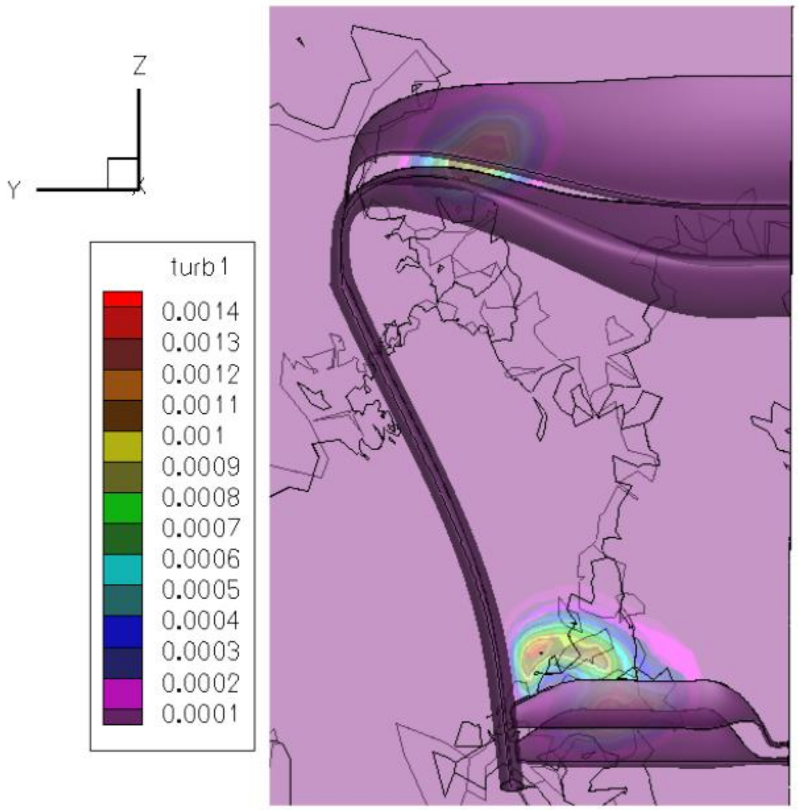


Figure 11: Rear wing geometry overlayed on wake cut planes 0.1 meters from the trailing edge of the rear wing with turbulent kinetic energy data: (a) 2021 F1 rear wing; (b) 2022 F1 rear wing.

The turbulent kinetic energy is analyzed through multiple cut planes in the wake of each wing in Figure 12. The y-axis represents a normalized turbulent kinetic energy termed $\hat{k}$ while the x-axis is the distance in meters. The following equation is used to calculate $\hat{k}$:

$$\hat{k} = \frac{\int_A k}{\int_{A_{inflow}} k}$$

A preliminary overview reveals that the 2021 wing generated less turbulence than the 2022 design, contrary to the intentions of the new technical regulations. At its peak, the turbulent kinetic energy of the 2021 wing is 18% lower than that of the 2022 wing. At one car length behind the wing, a car following at this distance would experience 125% more turbulence behind the 2022 rear wing compared to the 2021 wing.

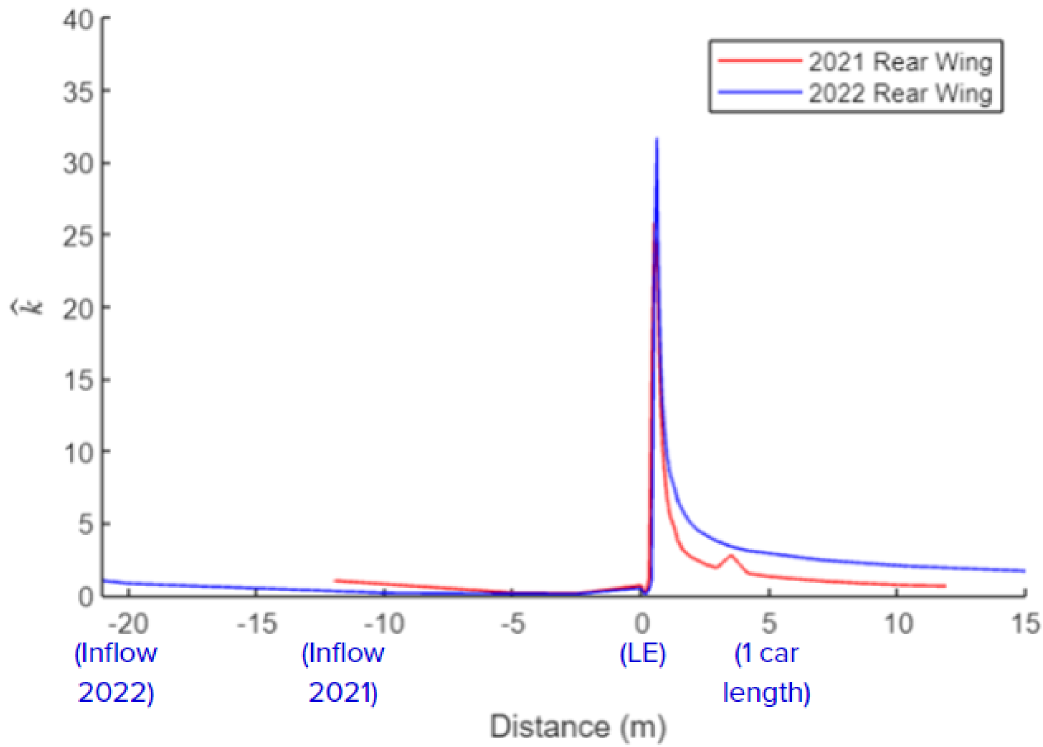


Figure 12: Comparison of the normalized turbulent kinetic energy $\hat{k}$ for the F1 2021 and 2022 rear wings.

Focusing on the wake region of the turbulent kinetic energy plot in Figure 13, it is observed that the turbulence in the wake of the 2021 wing increased and dissipated earlier than the turbulence generated by the 2022 wing. However, the turbulence in the wake of both rear wing designs appears to dissipate at approximately the same rate.

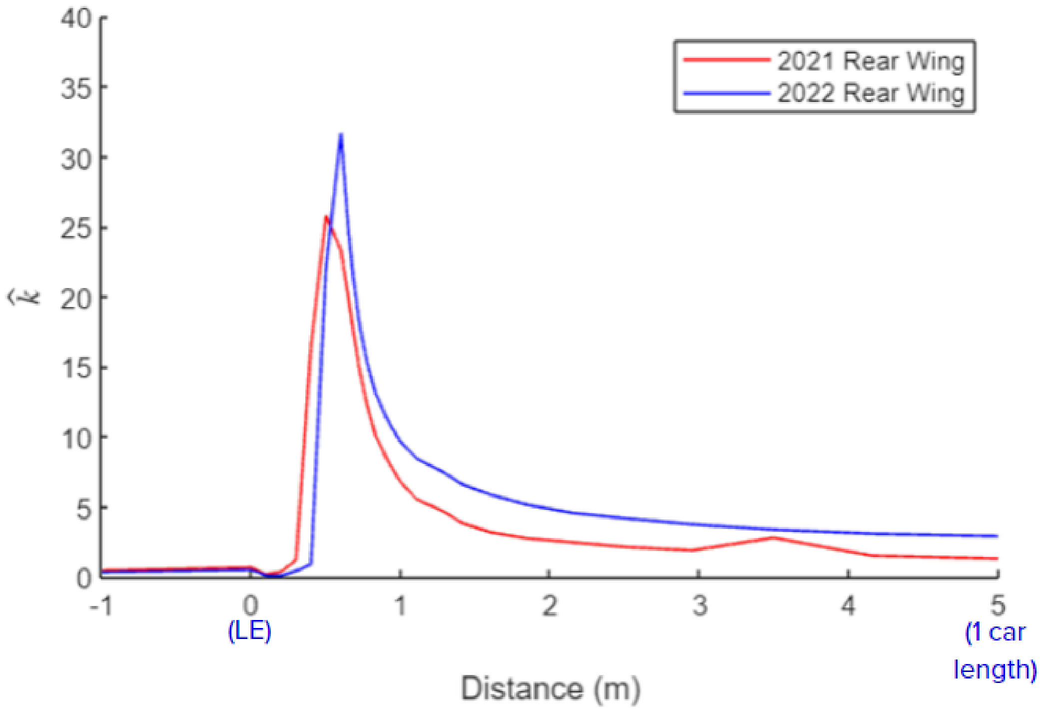


Figure 13: Comparison of the normalized turbulent kinetic energy $\hat{k}$ in the wake of the F1 2021 and 2022 rear wings.

While there are currently no statistics that outline the practical impact of the 2022 F1 technical regulations on the turbulence in the wake, there was a significant increase in overtakes in 2022 compared to 2021. According to Pirelli, the official tire provider for F1, there were 599 overtakes in 2021 and 785 overtakes in 2022 over the same 22-race span [22]. This 30% increase in overtakes likely indicates that the technical regulations are fulfilling their stated goal of increasing the competitiveness of the sport. Whether this impact is a direct result of the aerodynamics rule change is unclear, but it is highly probable as aerodynamics were a major focus of the new technical regulations. As such, the results of the current study contradict reality.

The discrepancy between the findings of the current study and the positive impact the regulation changes had in F1 during the 2022 season likely results from several factors that influence the study. Firstly, the choice of turbulent kinetic energy as a metric for quantifying turbulence might not adequately capture its characteristics. Additionally, the simulation modeled the rear wings floating in a volume without considering the rest of the car. As noted earlier, the 2022 rear wing operates in conjunction with the diffuser to funnel and lift the turbulence above a trailing car. Moreover, by not modeling the whole car, many other turbulence generating aerodynamic features, which have a definite impact on the wake of the car, are omitted. Since the technical regulation changes impacted every aspect of the aerodynamics of the car, the full impact of the turbulence in the wake could only be truly observed by modeling the car in its entirety. Furthermore, the ground introduces further complexity that would alter the flow. While the ground may not have a large impact on the rear wing alone, omitting it likely still had an impact on the outcome of the models.

Furthermore, the simplified geometry used in the study, while resembling the real wings, could have impacted the outcome. As previously discussed, actual F1 rear wing geometry often includes complex serrations and slits to improve the efficiency of the wing, especially on the 2021 rear wing [23]. These serrations and slits disrupt the airflow over the rear wing, impacting the turbulence in the wake. By not modeling these intricacies, the model is not a faithful representation of the turbulence in the wake of an F1 car.

4 | Conclusion

The intent of the current study was to establish whether the 2022 Formula One Technical Regulations were effective at reducing the turbulence in the wake of the car, thereby making it easier to make on-track overtakes and improving the competitiveness of the sport. The study utilized CFD to observe the turbulent kinetic energy at varying distances behind an F1 2022 rear wing and 2021 rear wing for the purpose of making a direct comparison between the two.

The results obtained in the current study suggest that the 2022 technical regulations fail to reduce turbulence in the wake of the car. The peak turbulent kinetic energy of the 2021 wing is 18% lower than that of the 2022 wing. However, the turbulence dissipation rate is the same for both wings.

If the study were to be redone, the model would include the full F1 car and the ground. Additional details would be included to make the geometric representations more accurate to the real-world counterparts. Additionally, other methods of quantifying turbulence would be investigated, and the solution would be run on the Ohio Supercomputer to accommodate for the greater geometric detail and a finer mesh resolution.

Bibliography

[1] B. Sullivan, “Introduction to CFD for F1,” Formula Uno Analisi Tecnica, https://www.funoanalisitecnica.com/2021/09/f1-cfd-navier-stokes.html (accessed Apr. 14, 2023).

[2] M. ur Rehman, “How Ground Effect Work in Formula 1,” Formulapedia, https://formulapedia.com/ground-effects-f1/ (accessed Jun. 19, 2023).

[3] B. Iskander, “Computational fluid dynamics in motorsports - Quo Vadis?,” Racecar Engineering, 16-May-2017. [Online]. Available: https://www.racecar-engineering.com/advertisement/fluid-dynamics-in-motorsport/#:~:text=CFD%20in%20Motorsports&text=In%20the%20early%201990s%2C%20two,increasing%20availability%20of%20computational%20resources. (accessed: 11-Mar-2022).

[4] U. Ravelli and M. Savini, “Aerodynamic investigation of blunt and slender bodies in ground effect using openfoam,” International Journal of Aerodynamics, vol. 7, no. 1, p. 36, May 2020. doi:10.1504/ijad.2020.107161

[5] U. Ravelli and M. Savini, “Aerodynamic simulation of a 2017 F1 car with open-source CFD code,” Journal of Traffic and Transportation Engineering, vol. 6, no. 4, 2018. doi:10.17265/2328-2142/2018.04.001

[6] J. Newbon, D. Sims-Williams, and R. Dominy, “Aerodynamic analysis of Grand Prix cars operating in wake flows,” SAE International Journal of Passenger Cars - Mechanical Systems, vol. 10, no. 1, pp. 318–329, 2017. doi:10.4271/2017-01-1546

[7] R. Perry and D. Marshall, “An evaluation of proposed Formula 1 aerodynamic regulations changes using computational fluid dynamics,” 26th AIAA Applied Aerodynamics Conference, Aug. 2008. doi:10.2514/6.2008-6733

[8] “2022 Formula One Technical Regulations.” Fédération Internationale de l'Automobile, 19-Feb-2021.

[9] “2021 Formula One Technical Regulations.” Fédération Internationale de l'Automobile, 16-Dec-2020.

[10] G. Piola and M. Somerfield, “The design trends to watch in the 2022 F1 car launches,” Motorsport.com, https://www.motorsport.com/f1/news/the-design-trends-to-watch-at-the-2022-f1-car-launches/7957648/ (accessed May 18, 2022).

[11] “New rear wing used by Charles Leclerc in Canada was initially planned for Silverstone,” Scuderia Fans, https://scuderiafans.com/f1-ferrari-new-rear-wing-used-by-charles-leclerc-in-canada-was-initially-planned-for-silverstone/ (accessed Sep. 16, 2022).

[12] M. Scott, "Red Bull Delight as FIA introduce new Rear Wing Test in Qatar," PlanetF1, https://www.planetf1.com/news/fia-new-rear-wing-test/ (accessed Jan. 20, 2023).

[13] N. Ashton, A. West, S. Lardeau, and A. Revell, "Assessment of rans and DES methods for realistic automotive models," Computers & Fluids, vol. 128, pp. 1–15, 2016. doi:10.1016/j.compfluid.2016.01.008

[14] A. Heft, T. Indinger, and N. Adams, "Experimental and numerical investigation of the DRIVAER model," In Proceedings of the ASME 2012 Fluids Engineering Division Summer Meeting collocated with the ASME 2012 Heat Transfer Summer Conference and the ASME 2012 10th International Conference on Nanochannels, Microchannels, and Minichannels., vol. 1: Symposia, Parts A and B, pp. 41–51, 2012. doi:10.1115/fedsm2012-72272

[15] B. Arrondeau, A. Saravana, A. Sabatés, and S. Daniela, Front Wing Design of a 2021 F1 Race Car, 2020.

[16] N. Simmonds *et al.*, "Complete body aerodynamic study of three vehicles," *SAE Technical Paper Series*, Mar. 2017. doi:10.4271/2017-01-1529

[17] X. Castro and Z. A. Rana, "Aerodynamic and structural design of a 2022 formula One front wing assembly," MDPI, vol. 5, no. 4, p. 237, 2020.

[18] "Average F1 car speed – we pulled the real data (2023)," Professionals HQ, https://professionalshq.com/average-f1-car-speed-we-pulled-the-real-data-2023/ (accessed Mar. 8, 2023).

[19] Username: Latios, "Aero analysis for F1 2022 based on CFD result," F1technical, https://www.f1technical.net/forum/viewtopic.php?t=30329 (accessed Feb. 10, 2023).

[20] A. Ono, "Slipstream and 'dirty air' explained," Racecar Engineering, https://www.racecar-engineering.com/tech-explained/slipstream-and-dirty-air-explained/ (accessed Jun. 24, 2023).

[21] S. Mansell, *Why Formula 1's New CURVED Wing Is GENIUS*. Driver61, 2021.

[22] L. Smith, "Pirelli reveals 30% increase in F1 overtakes from 2021 to 2022," Motorsport.com, https://us.motorsport.com/f1/news/pirelli-reveals-30-increase-in-f1-overtakes-from-2021-to-2022/10403865/ (accessed Jul. 15, 2023).

[23] M. Somerfield and G. Piola, "Tech Analysis: Why serrated wings have made an F1 return," Motorsport.com, https://us.motorsport.com/f1/news/tech-analysis-why-serrated-wings-have-made-an-f1-return-678135/2971368/ (accessed Jun. 24, 2023).

[24] A. Guerrero and R. Castilla, “Aerodynamic study of the wake effects on a formula 1 car,” Energies, vol. 13, no. 19, p. 5183, Oct. 2020. doi:10.3390/en13195183

[25] “FUN3D manual :: Chapter 1: Overview and Getting Started,” NASA. [Online]. Available: https://fun3d.larc.nasa.gov/chapter-1.html. [Accessed: 11-Mar-2022].

[26] J. F. Dannenhoffer and B. Haimes, “ESP Overview & Getting Started.” Jun-2021.

[27] Racecar Engineering, “Tech explained: 2022 F1 technical regulations,” Racecar Engineering, https://www.racecar-engineering.com/articles/tech-explained-2022-f1-technical-regulations/ (accessed Jun. 24, 2023).