Formula One Car Downforce Analysis

Jordan T. Smith[1]
Mississippi State University, Starkville, MS, 39759, USA

This document entails the background and process for conducting a project geared towards analyzing Formula One car front wings. The design process includes creating a three-dimensional CAD rendering of three Formula One cars and using CFD to analyze their performance via downforce at various speeds. A comparison of the measured downforce levels is compared to real-world data using Amazon Web Service data collection and video coverage from a selected event. Results indicate that wing geometry has a significant effect on its respective performance.

Nomenclature

CAD   = Computer Aided Design
CFD   = Computational Fluid Dynamics
F1   = Formula One
FIA   = Federation Internationale de l'Automobile
Mph   = Miles Per Hour
Spa   = Spa de-Francorchamps

I. Introduction

The sport of Formula One exudes perfection and attention to detail in every manner. Every aspect of the Formula One race car is crafted to increase speed and performance, while decreasing weight and drag. Factors such as driver weight, fuel level, tire pressure, etc. are carried to the fourth and even fifth significant digit. From an aerodynamic standpoint, the largest areas of focus for the Formula One car are two control surfaces, the front and rear wing. The purpose of each control surface is to create downforce to allow the car and driver to corner faster. Each race weekend and change in track causes a team to have a different aerodynamic attack. Different circuits require different drag to downforce ratios and therefore different front wing geometries. The complexity of the front wing's geometry changes depending on the downforce level for each track. The ability of a team to maximize the drag to downforce relationship in their favor, ultimately is what separates the race winner from the others. The aerodynamic development of front wings for a given race's weekend is critical to team performance. To best understand the effect geometry has on front wing performance, a test concerning three teams across one event is required. Through analysis of the Mercedes AMG, Scuderia Ferrari, and Aston Martin Red Bull Racing teams' front wings at the 2020 Belgian Grand Prix, the engineering behind the tradeoff of downforce and drag can be highlighted. CAD models will be created through the SolidWorks 2017 program by Dassault Systems. Following the creation of the CAD models, loads will be computed through SolidWorks' flow simulation tool. Due to the high amount of confidential information, most Formula One teams will not release any performance data to the general public. In the scope of this project, Amazon Web Services data and event commentary will be used to assist with result comparisons.

II. Design Process

A. Problem Statement

The 2020 Formula One regulations saw a simplification from earlier years front wing geometries, in an effort to create more competitive racing and reduce costs on teams. The intent of this project is to research 3 Formula One team's front wings, Mercedes, Ferrari, and Red Bull. To keep variables to a minimum, each car's front wing will be analyzed around the same circuit, Spa de-Francorchamps. This research will be performed through Computational Fluid Dynamics in a SolidWorks program.

---

[1] MSU Undergraduate Student, Aerospace Engineering, and AIAA Member.

B. Objectives

The objectives for this project focus around analyzing the front wings at various criteria. Each wing will be analyzed at three speeds. The first speed is representative of a high-speed portion of the track named the Kemmel Straight (velocity of approximately 200 to 212 mph). The purpose of analyzing the high-speed section is to best determine the drag of each wing. The second speed is representative of a medium-speed corner where downforce is most crucial as the downforce in these corners allows the cars to go quicker. Finally, the last speed is representative of a low-speed corner to see how a slower airflow will react with the front wings and cause possible turbulent airflow. The final objective of this project is to compare the results acquired through experimentation to known accepted values from the current Formula One season. Due to the fact that teams do not release empirical data to the public, these accepted values will be determined by quantitative and qualitative analysis of the 2020 Belgian Grand Prix broadcast and any data Amazon Web Services releasees to the public.

C. Background Research

Two articles written by Formula One gearheads were read and analyzed to give more background and information on the subject. Overall, the message is clear, Formula One regulations for the current race season have an extremely large impact on the performance of each car's aerodynamics and generally teams with larger budgets can improve their aerodynamics to a point of invulnerability. In addition, the results of aerodynamic studies for the current Formula One season may be skewed due to the technology available to drivers in the current cars. A large technological learning curve occurred when hybrid engines were introduced in 2014. These hybrid engines have allotted drivers with systems such as a battery boost that makes ordinary aerodynamic optimization obsolete. It was nice to review these articles to allow for a decrease in bias once the 2020 Formula One season car's specifications and data was obtained.

III. Preliminary Set-up

A. Mercedes

Initially, the Mercedes components proved to be the simplest to replicate in a CAD model. In addition to having a large amount of community made models, the Mercedes had the best pictures to determine the geometry of the vehicle's front wing. Detailed below is a front view picture of the 2020 Mercedes Formula One car at Spa. This picture was used frequently when creating the respective CAD model.

Figure 1 Mercedes 2020 Belgian Grand Prix Front Wing

In addition to using photos to replicate each wing in SolidWorks, the released Formula One technical regulations were a huge help in assisting with the overall dimensions of each CAD shape. These regulations allowed for a quantitative property to be associated with the observed wing geometry. The main challenge while creating the CAD files was adjusting the depth of each component to best reflect the expected geometry. This concept is expected to be one of the leading causes of experiment error.

B. Ferrari and Red Bull

The process for the Ferrari and Red Bull front wings was extremely like that of the Mercedes. The below figures were analyzed and replicated as accurately as possible. All front wings were created without the front stalk that connects the wing to the subsequent parts of the car. This was excluded for many reasons; however, the two main reasons were due to the complicated nature of that component and the chance to skew results by adding another component to analyze. In addition, this allows for the CFD analysis to focus and isolate on the actual front wing, endplates, and small wing slats.

Figure 2 Ferrari 2020 Belgian Grand Prix Front Wing

Figure 3 Red Bull 2020 Belgian Grand Prix Front Wing

IV. Experiment and Expected Values

The components of the experiment echo the requirements detailed by the objectives of this experiment. Following the completion of the CAD models in SolidWorks, the models are subjected to the SolidWorks flow simulation at three different speeds: 93.47 meters per second, 49.174 meters per second, and 24.587 meters per second. These velocities equate to 210, 110, and 55 miles per hour. The SolidWorks flow simulator was set to globally solve for forces (in Newtons) in each direction. In addition, the flow trajectories were analyzed to qualitatively observe the flow characteristics of each team's optimal geometry.

Ideally this experiment would be conducted physically in addition to electronically. However, wind tunnel testing is ultimately not optimal for this experiment due to model scaling and therefore an increase in fluid velocity to maintain a consistent Reynold's number. While there are no published values to give a good estimate of expected values, quantitative analysis of the 2020 Belgian Grand Prix lends the hypothesis that the Red Bull wing should create the most downforce and the Ferrari creating the least. Another factor of interest was the calculated drag values. While drag is not the focus of this experiment, it is interesting to determine how much the downforce costs the teams in other performance areas.

V. Results

Following 9 iterations of the SolidWorks flow simulator; drag, downforce, and side force values were calculated. In addition, the flow trajectories were calculated and the flow over and around the surfaces was observed. These speeds were set at the same atmospheric pressure (101,325 Pascals) and temperature (288.15 K).

Team Name	93.47 meters per second	49.17 meters per second	24.59 meters per second
Mercedes	1195.12 N	329.476 N	82.445 N
Red Bull	1463.403 N	394.127 N	99.389 N
Ferrari	861.9 N	233.417 N	58.19 N

Figure 4 Table Displaying Downforce Values (Newtons)

The results were conclusive with the expected results. All force values in the above tables are averages. The Red Bull wing created the highest amount of downforce, and when observing the flow trajectories (pictured below) the Red Bull wing created vortices only around the endplate region.

A. High-Speed

Analysis of the high-speed performance of each wing details the hypothesis that the performance will be adjusted due to the geometry of the front wing design.

Team Name	93.47 meters per second					
	Average		Maximum		Minimum	
	Downforce	Drag	Downforce	Drag	Downforce	Drag
Mercedes	1195.12 N	1029.916 N	1197.160 N	1030.874 N	1192.982 N	1028.883 N
Red Bull	1463.403 N	1211.764 N	1469.783 N	1218.867 N	1455.536 N	1206.562 N
Ferrari	861.9 N	998.714 N	866.163 N	1000.3 N	856.362 N	997.74 N

Figure 5 Table Displaying High-Speed Downforce/Drag Values

The drag force components were added to illustrate the intricate relationship of downforce to drag. As highlighted by the data, the Red Bull wing has the highest downforce at the cost of the highest drag. The Ferrari wing by comparison creates less downforce than drag. A concept that is not ideal in the automotive racing profession and can be attributed to poor CAD design by the experimenter. The figures below illustrate the flow profile over each wing.
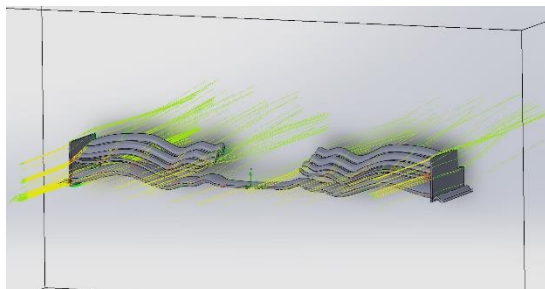

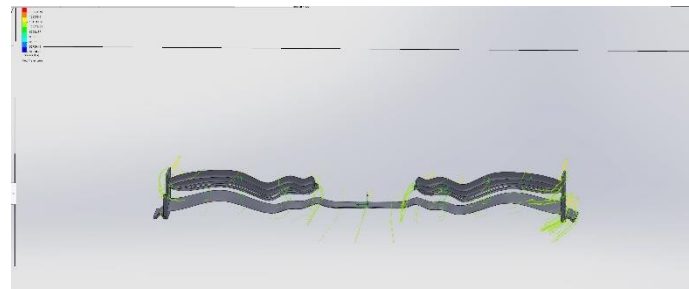**Figure 6 Front View of Flow Trajectory Ferrari**

Figure 7 Rear View of Flow Trajectory Ferrari

The Ferrari's wing flow at high-speed show multiple vortices at the boundary of the small wing slats and the endplates at the edges of the wing. These vortices give an explanation at the massive performance detriments the Ferrari wing experienced. Additionally, it is important to note that most of the pressure is distributed around 102,000 – 103,000 Pascals.
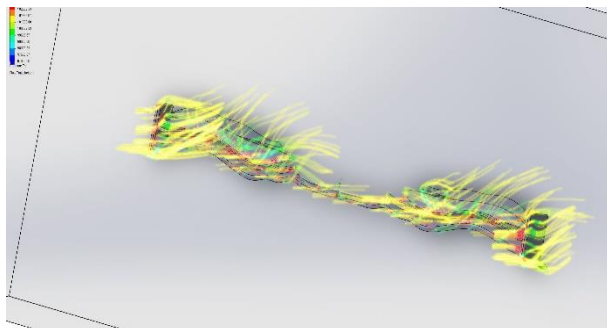

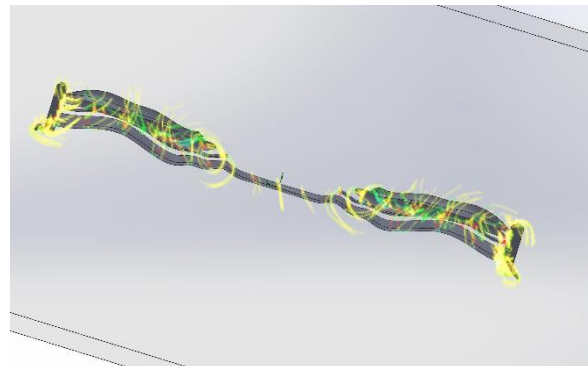**Figure 8 Front View of Flow Trajectory Mercedes**

Figure 9 Front View of Flow Trajectory Mercedes

Like the Ferrari wing, the Mercedes wing had a large pressure distribution between the directionally front and back of the front wing slats. Additionally, the Mercedes wing has more pronounced vortices at its boundaries which gives a hypothesis to the increased performance of this wing. Additionally, the geometry of the Mercedes (and Red Bull)

wing(s) is much simpler than the Ferrari. Less pieces and gaps between these pieces would limit airflow and potentially control the air around the contours more. This leads to not only increased drag, but also increased downforce.
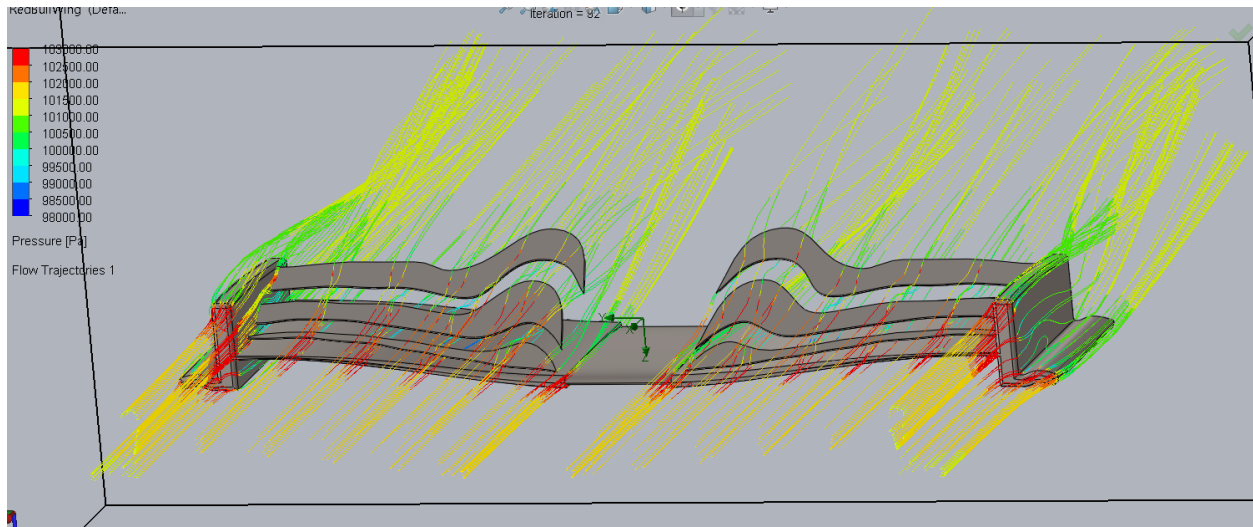

Figure 10 Front View Flow Trajectory Red Bull

The flow depicted in Figure 10 best illustrates the vortex effect and the pressure distribution between the front and rear faces of the three wing slats. The flow in the middle of these slats is nearly perfectly straight, this offers a benefit to car performance as the air glides perfectly to the front tires. This has multiple effects, most notably, cooling the tire and creating additional Newtons of downforce. When compared amongst each other it is easy to discern that at high speeds, wing geometry has a direct correlation to wing performance.

B. Mid-Speed

Analysis of the mid-speed performance of each wing further details the hypothesis that the performance will be adjusted due to the geometry of the front wing design.

| Team Name | 49.174 meters per second | | | | | |
| | Average | | Maximum | | Minimum | |
	Downforce	Drag	Downforce	Drag	Downforce	Drag
Mercedes	329.476 N	287.163 N	330.050 N	287.661 N	328.611 N	286.778 N
Red Bull	394.129 N	333.221 N	396.535 N	334.784 N	392.933 N	332.258 N
Ferrari	233.417 N	272.836 N	234.549 N	273.339 N	231.741 N	272.381 N

Figure 11 Table Displaying Mid-Speed Drag/Downforce Values

The mid-speed results were extremely like the high-speed results. A minor difference is the Mercedes' gap between downforce value to drag value is consistent with the high-speed experiment. Meanwhile, the Ferrari and Red Bull have seen performance drop; the difference between downforce and drag at the mid-speed experiment is smaller than that of the high-speed test.
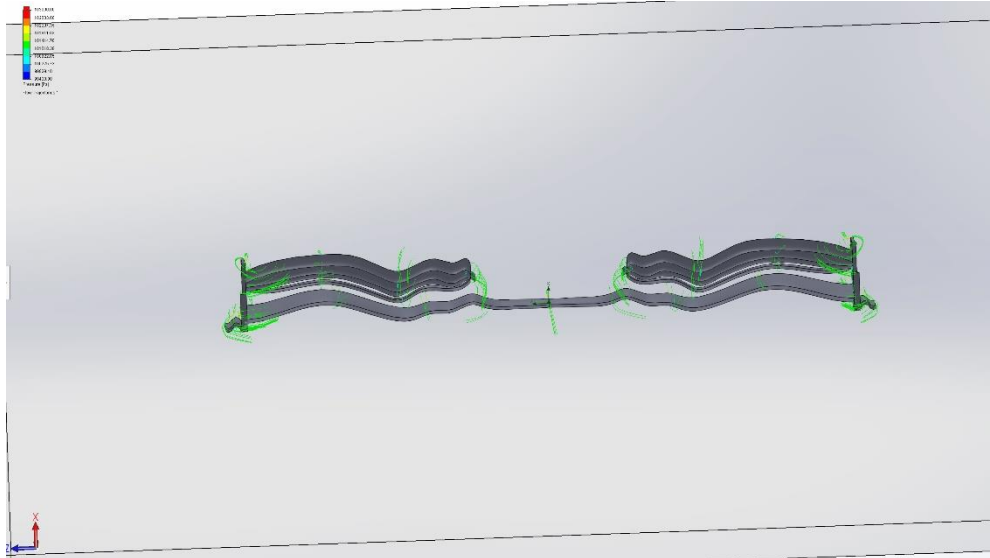
Figure 12 Rear View Mid-Speed Flow Trajectory Ferrari

In comparison to the high-speed flow trajectories, the mid-speed Ferrari flow constructs a similar profile. The biggest difference is the pressure gradients on the surfaces are much lower, centered around 101,500 Pascals, as the flow has slowed.
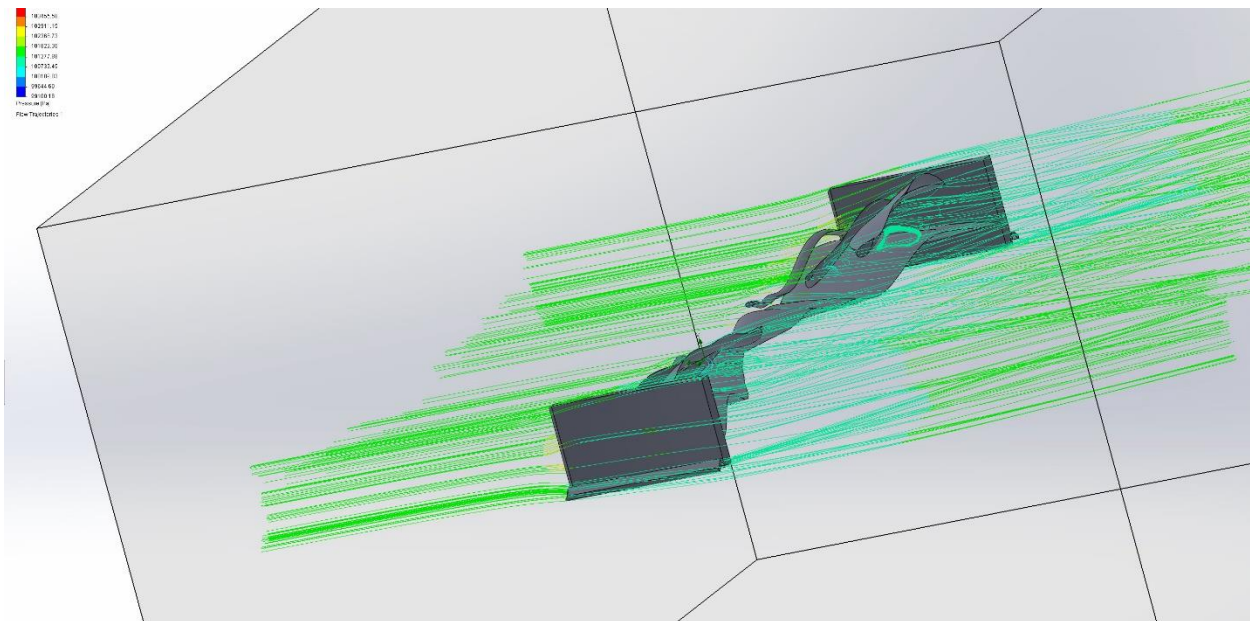

Figure 13 Side View Mid-Speed Flow Trajectory Mercedes

The Mercedes wing follows the trend of lower pressure gradients. An area of note is the large vortex underneath the far wing slats. This swirling air is likely due to an incorrect sketch of its geometry, as most engineers/designers would not look to develop a large energy loss in such a location. Additionally, this and subsequent figures, illustrate the higher pressure on the front (figure's left) of the wing and a lower pressure on the rear (figure's right). These pressures are around 102,000 Pascals.
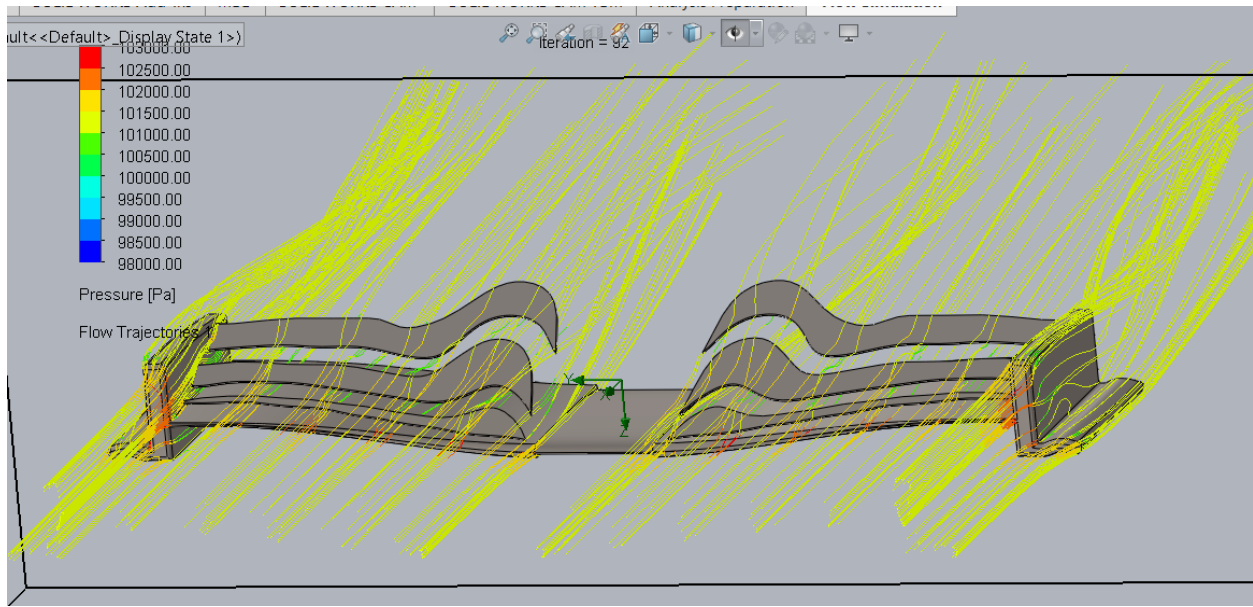
Figure 14 Front View Mid-Speed Flow Trajectory Red Bull

C. Low-Speed

Analysis of the mid-speed performance of each wing further details the hypothesis that the performance will be adjusted due to the geometry of the front wing design.

| Team Name | 24.587 meters per second | | | | | |
| | Average | | Maximum | | Minimum | |
	Downforce	Drag	Downforce	Drag	Downforce	Drag
Mercedes	82.445 N	71.938 N	82.757 N	72.059 N	82.250 N	71.814 N
Red Bull	99.389 N	83.258 N	100.182 N	83.573 N	98.971 N	83.070 N
Ferrari	58.191 N	68.358 N	58.480 N	68.447 N	57.846 N	68.283 N

Figure 15 Table Displaying Low-Speed Drag/Downforce Values

Analysis of the low-speed results assist to prove the correlation between geometry and performance, in addition to speed and force created. An interesting note for optimization is the drastic change between force levels of the high and low speeds. The Mercedes for instance lost nearly 1,000 Newtons of downforce, and a similar amount of drag. This highlights how important it is to weigh benefits and costs of adding/subtracting downforce. The subsequent figures visualize the flow at low speeds for each team's concept.
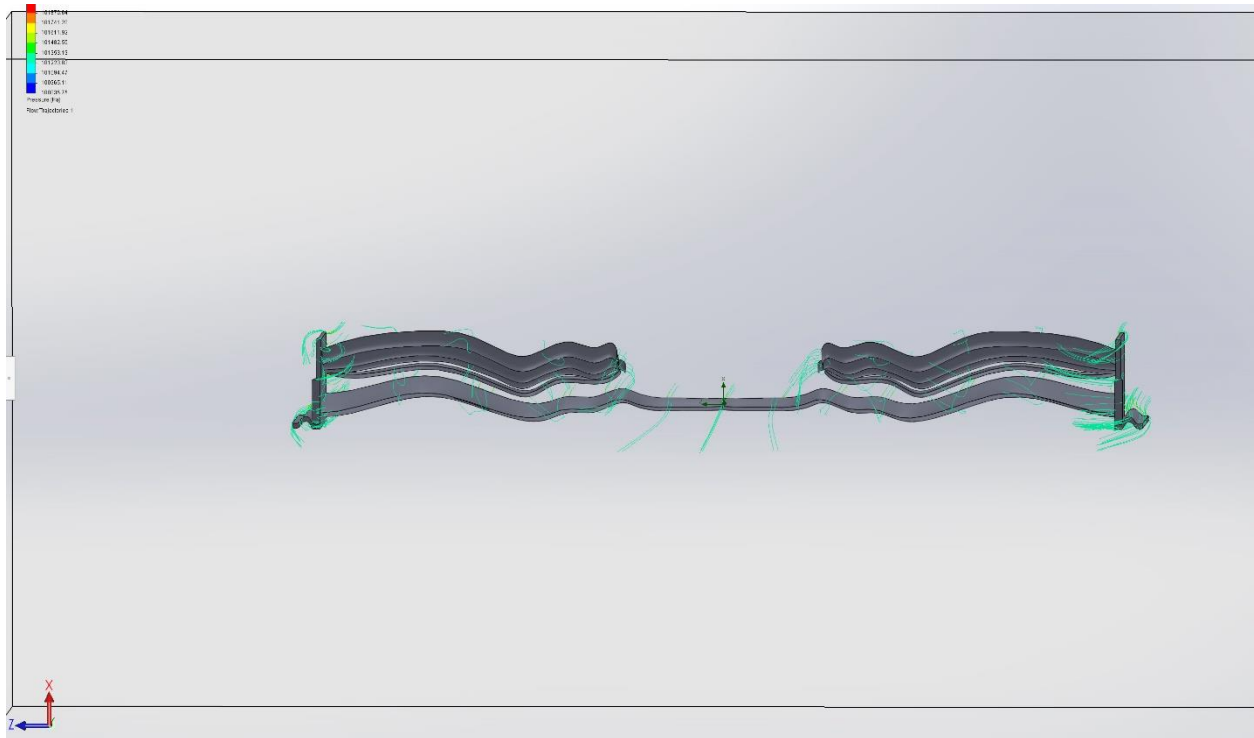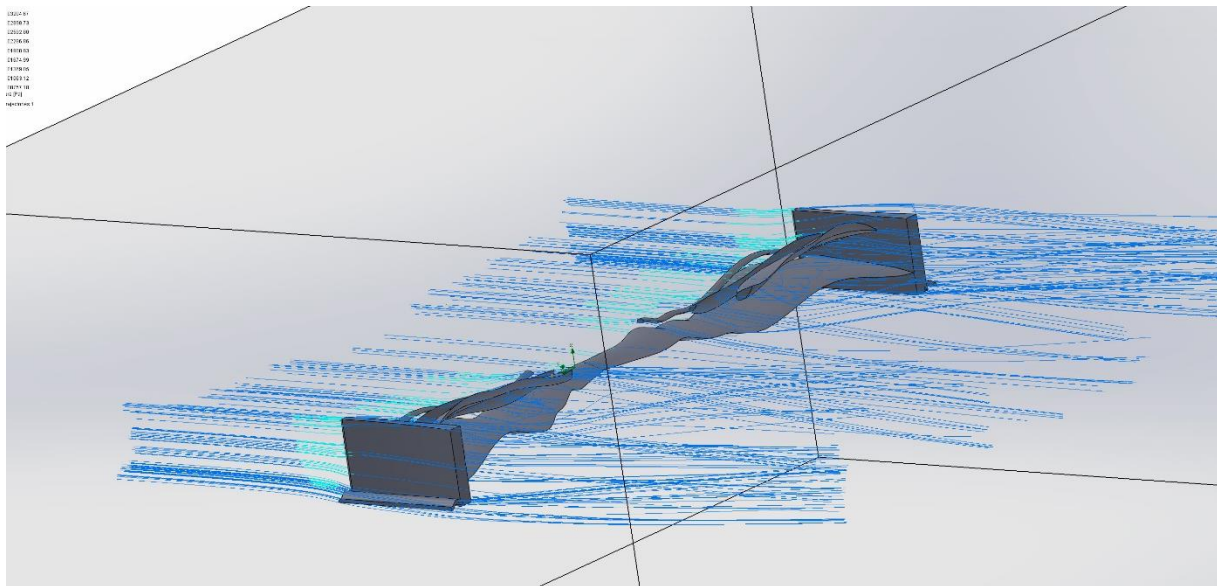
Figure 16 Rear View Low-Speed Flow Trajectory Ferrari

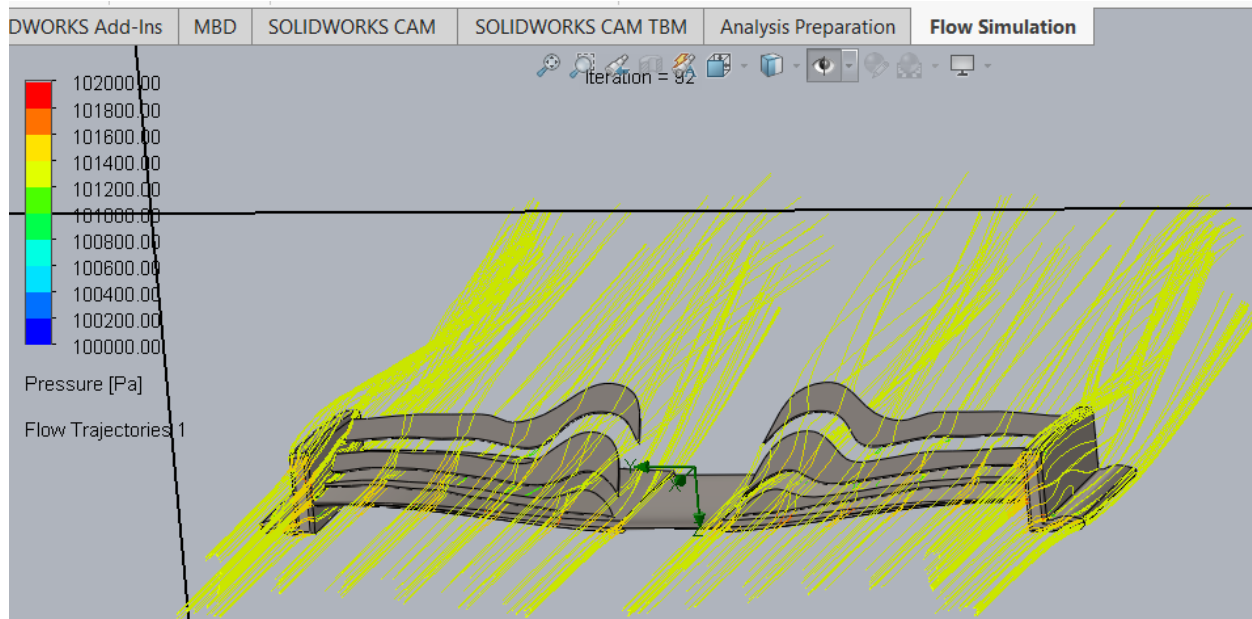**Figure 17 Side View Low-Speed Flow Trajectory Mercedes**

Figure 18 Front View Low-Speed Flow Trajectory Red Bull

Nearly all the flow distribution pressures at the low-speed experiment were at or near the set atmospheric pressure of 101,325 Pascals. The lower the amount of air moving over the surface at a given time changes the force created by the surface. In a practical race setting, the low-speed corners are where the mechanical grip of the tires helps the car turn better as opposed to the aerodynamic grip which is extremely helpful at the mid and high-speed corners. All flow trajectories, experiments, and values follow the hypothesis finding the correlation between wing geometry, performance, and downforce to drag.

VI. Conclusion

Through analysis of three Formula One wings at 3 distinct speeds, the direct relationship between wing geometry (most notable wing angles) and induced downforce/drag can be explored. Error induced by the design process can influence the exact precision of the results, however, the estimations presented by these figures give good insight into the explored concept.

Using and manipulating each front wing has easily been the most difficult part of this project. Even though the wing geometry is greatly simplified compared to previous years, each half of the wing contains about 6 to 7 sub-wings. With no true guide to wing angle and length, the easiest way to mimic the Belgian Grand Prix is to visually approximate. Obviously, this introduces a large amount of error as the approach turns from a scientific approach to more of an estimation. To combat this, each model has been cross referenced as much as possible with a technical regulation document from the Formula One governing body (FIA). Any discrepancy of data and results is mostly dedicated to this concept.

VII. Acknowledgments

Jordan Smith would like to acknowledge and thank his advisors Dr. Adrian Sescu and Mr. Calvin Walker for their continued support and phenomenal guidance. Additional acknowledgments include Mr. Rob Wolz for assistance with setting up and the project and updates on the wind tunnel. Finally, Mr. Cole Day gave a multitude of expertise with interfacing with the SolidWorks program. In addition, he assisted with double checking the geometry of each front wing, most notably the Red Bull wing.

VIII. References

[1]Anderson, Gary."Gary Anderson's Guide to Mastering Spa." *The Race*, 28 Aug. 2020, the-race.com/formula-1/gary-andersons-guide-to-mastering-spa/.

[2]Toet, Willem. "Willem Toet Explains...the 2019 F1 Aerodynamic Dilemma." *Race Tech Magazine Willem Toet Explains the 2019 F1 Aerodynamic Dilemma Comments*, www.racetechmag.com/2019/03/willem-toet-explains-the-2019-f1-aerodynamics-dilemma/.