Informal and Honest Documentation
WR-220c & WR-220e Aero Package

Written by Dylan Studden

Email: studden@go.uwracing.com
Phone: (218) 428-8435

“The best thing a human being can do is help another human being know more.” — Charlie Munger

Table of Contents

Personal Introduction

My first car was the WR-217c, where I was an averagely committed powertrain subteam member. The following season I completed a successful complete overhaul of the powertrain cooling system for the WR-218c. Then, I became the powertrain lead for the WR-219c. During that season, we burned the basement of ECB down, had to fight tooth and nail with the University to not kill the team for the rest of the year, only had 1 successful test day for the whole season, showed up late to competition, and got 70th overall at FSAE Michigan 2019. The best news coming out of the 219 season was that there was no possible way that the WR-220 season could be any worse.

The 2019-2020 season was to be my and Rishi's fourth and last year on Wisconsin Racing. After getting trounced in design prelims in vehicle dynamics and aero, Rishi and I decided to fill those roles respectively for the following year. Although we had next to zero knowledge on these topics at the end of the 219 season, we trusted that we could figure it out. I had never done CFD before, I asked previous aero leads very little questions about aero early on, and had next to zero experience with composite manufacturing processes. Although I did a fair bit of research and spent every free moment of my fall semester figuring out aero things, I was still very underprepared to take on this project. I admit that I did not do much of this perfectly, and I want to document the things I do know, and the things that I should have considered during this project. I'll try to mention aspects of my designs that are potentially wrong or things that I think may need to be researched/considered as I walk through my designs and processes in this report.

One of the most dangerous things you can do as a member for Wisconsin Racing is assume that the person who did this project before you knew what they were doing. Whenever you are taking over a project or iterating on last year's design, you should always be questioning the decisions of the member who had this project before you.

Impossibly, the 220 season was even worse than 219. Our year was cut short by the coronavirus. The WR-220 aero package design is (mostly) complete, but it was never implemented or validated. I will discuss plans for manufacturing (minimally) and validation as well in this document.

Background

Aero 101: Race Car Aerodynamics at the Highest Level

To put it simply, the job of the aerodynamic system is to create downforce with minimal drag, and supplement brake and powertrain cooling. It achieves this through use of aerodynamic devices that are used to efficiency control air flow over and through the vehicle. Aerodynamic design includes determining high level design goals of how much downforce you want the package to make, what kind of front-to-rear downforce balance is optimal, how much drag is too much drag, how much air needs to flow through the radiators, brakes, engine bay, etc. Once these goals are set, then the aerodynamicists develop aero device geometry to achieve these goals.

The primary goal of the aero package is to produce as much downforce as efficiently as possible (minimizing drag and weight). This is typically done through the use of front, rear, and (sometimes) side wings, and undertrays/diffusers.

Going back to your high school physics class, you learned that friction force, $F = \mu \cdot N$ where F is friction force, μ is the coefficient of friction, and N is the force normal to the surface. In this application, the friction force is the amount of grip available to the tires, and N is the weight of the vehicle plus any vertical aerodynamic load. Therefore, more downforce = more grip. Again referencing your high school physics notes, Newton's second law is $F = m \cdot a$ Thus, as a vehicle's available grip (F) increases, so does its available capacity to accelerate. This means that as available grip increases (as long as the driver is capable of using it) the faster the car can go around the track. This is demonstrated in a Lapsim study comparing vehicle speed during the course of a lap with and without aero, shown in figure 1.

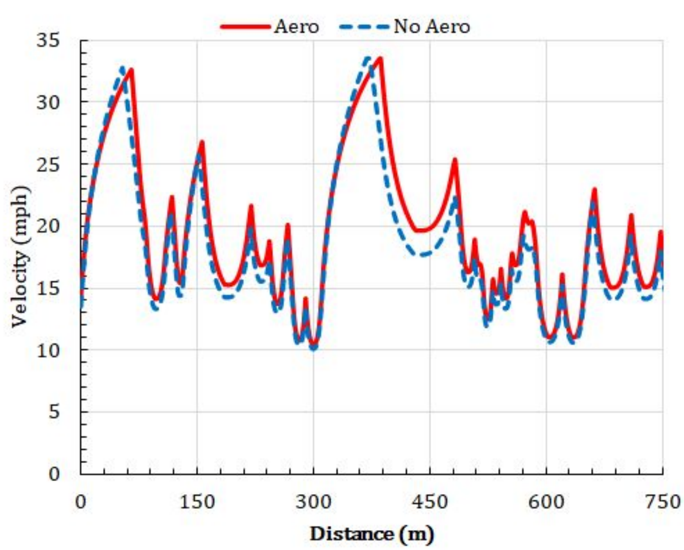


Figure 1: Vehicle velocity over the course of a lap.

Downforce comes at a cost, however. Adding aerodynamic devices will inherently induce some drag as well. This drag force opposes the force generated by the vehicle's powertrain, therefore slowing the car down. This drag force is also what limits the top speed of the vehicle.

Aero 102: Important Concepts

Lift and Drag

Drag and negative lift (downforce) are represented by very similar equations. The A used in both equation 1 and equation 2 corresponds to the **frontal area of the vehicle**. If you try to google what area to use for calculating lift vs. drag you will often see that you should use the frontal area for drag and the “top projected” area for lift. While this is the case for stand-alone wings and often aircrafts, this is not the case for road vehicles. The area used is arbitrary, but this convention makes it easier for us to compare drag/lift performance between cars and designs.

$$F_L = \frac{1}{2} \bullet \rho \bullet C_L \bullet A \bullet v^2 \quad \text{(equation 1)}$$
$$F_D = \frac{1}{2} \bullet \rho \bullet C_D \bullet A \bullet v^2 \quad \text{(equation 2)}$$

There are two sources of drag. Viscous drag and pressure drag. **Viscous drag** (also called **friction drag**), shown on the left in figure 2, is drag due to the friction between fluid particles and an object’s surface. From Newton’s Third Law we can understand that this friction that slows down flow will exert an equal and opposite force on the object. This force is what we refer to as friction drag. **Pressure drag** (also called **form drag**) is a result of a pressure difference across a surface. This is shown on the right in figure 2. Flow over the blunt body creates a high pressure in front of the body, and low pressure behind the body. This pressure difference times the area of the plate is equal to the force caused by pressure drag.

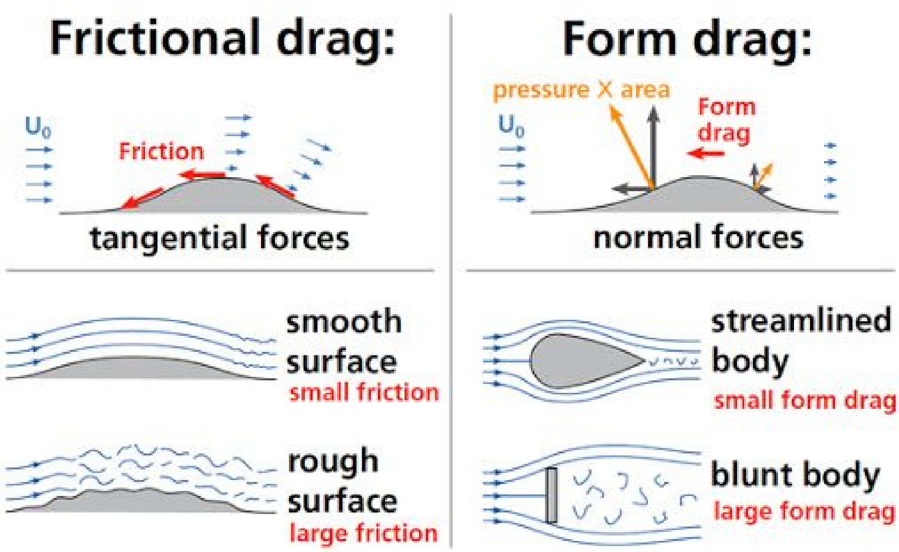


Figure 2: Different types of drag.

Center of Pressure

The **center of pressure** (CP) is *the point where the total sum of a pressure field acts on a body, causing a force to act through that point. The total force vector acting at the center of pressure is the value of the integrated vectorial pressure field.* Essentially, this means that all aerodynamic forces acting on a body can be modeled as a single force vector acting on a single point within that body. This single point is called the center of pressure.

Analogous to the center of pressure is the center of gravity (CG). All gravitational forces acting on a body can be modeled as one force vector acting on a single point. The concept of center of gravity is usually more intuitive to people than center of pressure, but the concept is equivalent for both.

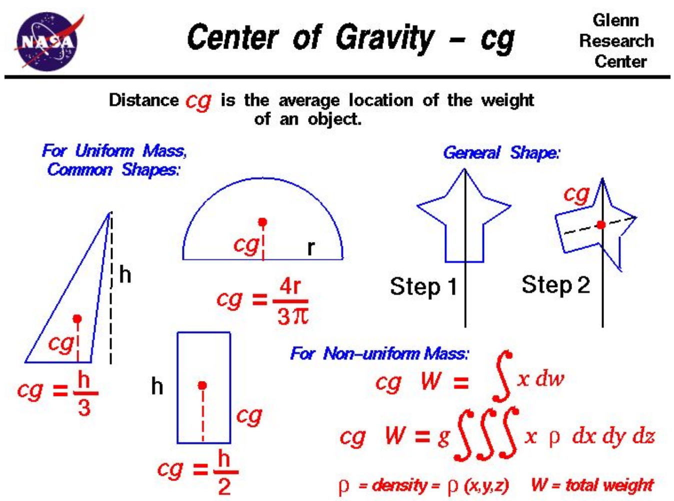


Figure 3: Center of gravity visualization.

There are essentially 2 main reasons that we care about the center of pressure in the design of our aero package: **stability** and **balance**.

Aerodynamic Stability

A dart (the kind that you throw at a dartboard) is an **aerodynamically stable** object. When a dart is thrown, it is initially a bit wobbly, before self correcting and stabilizing itself. This is because on a dart, the center of pressure is rearwards of the center of gravity. If a dart is thrown backwards (fins first), the center of pressure is in front of the center of gravity. The aerodynamic forces acting on the front of the dart will induce a moment about the CG, before forcing the dart to flip around and face point-forwards. This concept applies to race cars too.

As seen in figure 4, a vehicle with its **CP frontwards of the CG is aerodynamically unstable**, while a vehicle with its **CP rearwards of the CG is aerodynamically stable**. In an aerodynamically unstable car, the driver will essentially always be fighting with the air to keep the car going in a straight line.

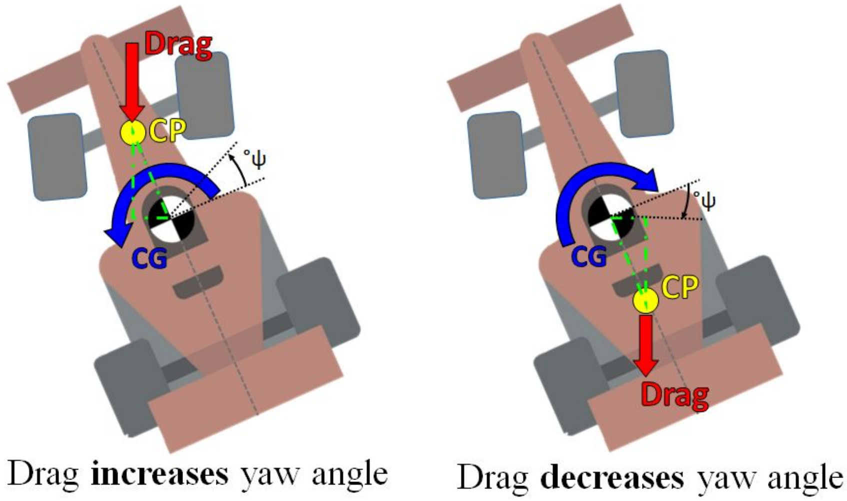


Figure 4: Effect of CP on vehicle yaw stability.

You may ask: “What about in a corner? Don’t you *want* a yaw moment when going through a turn?” Yes. Especially in FSAE, it is very important to be able to achieve high yaw rates for corning. Generally, you would want as much aerodynamic instability as your drivers can handle. This allows for maximum cornering performance, without sacrificing straight-line control.

The center of pressure location changes with vehicle speed. The rear wing receives relatively dirty air that comes off of the front wing, monocoque, driver, etc. At high flow velocity, fluids have a higher tendency to become turbulent. Interaction with objects like suspension members and regions like the cockpit can have greater effect on air turbulence at high speed. Therefore, air supply to the rear wing becomes dirtier at higher vehicle speeds. This effect causes the rear wing to decrease in efficiency as vehicle speed increases. Thus, **the CP moves forward the faster the car goes**. Assuming we never want the car to become aerodynamically unstable in standard trim, the corner case for checking CP location is at top speed.

Some events, however, do not have much of a need for straight line stability, like skidpad for example. This is why we tend to run a severe amount of negative rear toe in a skidpad setup - to induce yaw instability. The CP could be pushed forwards by trimming the rear wing down (decreasing flap angle - producing less rear downforce), and trimming the front wing up (increasing element angle - producing more front downforce). This could theoretically increase the yaw moment on the vehicle, decreasing skidpad times. Because skidpad is run at relatively low speeds, this could produce little noticeable effect. This effect could be determined through CFD, or just a thorough quick test in the fall.

Because our drivers are typically pretty inexperienced, I like to target a nominal CP location to be directly on top of/slightly rearwards of the CG at top speed. For the 220c&e I targeted the CP to be 5% more rearwards that the CG. Note that the CG locations are significantly different between the 220c and 220e.

Aerodynamic Balance

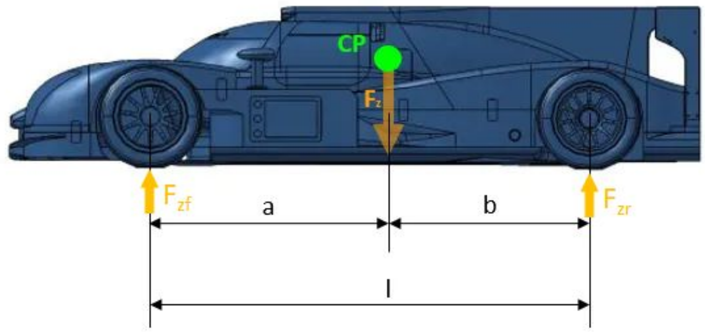


Figure 5: Front to rear downforce distribution is affected by CP location. It's really just a statics problem.

During medium to high speed turning and braking, the vertical aerodynamic load at each tire has a significant effect on available grip. A far forward CP will cause the rear tires to have less grip than the front, inducing *oversteer*, and a far rearwards CP will cause the front tires to have less grip than the rear, inducing *understeer*. To prevent aero-bias-induced oversteer or understeer, the front-to-rear CP location needs to be the same as the CG.

By placing the top speed CP slightly behind the CG the vehicle will always be aerodynamically stable but there will always be some aero-bias-induced understeer. The aero-bias-induced understeer can be compensated for through some vehicle dynamics/chassis setup black magic that I won't get into.

Pressure

Bernoulli's Principle is a relatively simple, yet very important concept to understand. This principle basically states that the pressure of a fluid can be broken up into two components: **static pressure** and **dynamic pressure**. You've observed static pressure when you drive down a mountain road and your ears pop as pressure equalizes across your ear canal. You are probably aware that atmospheric pressure is lower at high elevations than at sea level. This refers to static pressure. Dynamic pressure is the pressure you feel when you stick your hand out of the window of a moving car. This dynamic pressure is due to the relative velocity of the moving fluid. **Total pressure** of a fluid is static pressure plus dynamic pressure. Another way to think about pressure is the amount of energy in a fluid per unit volume.

$$P = \frac{Force}{Area} = \frac{F}{A} = \frac{F \cdot d}{A \cdot d} = \frac{W}{V} = \frac{Energy}{Volume}$$

Figure 6: This equation set demonstrates the relationship between pressure and energy.

It's ok if this doesn't really click with you, but just accept that it is true, and understand the math in figure 6. This is important in understanding the relationship between static and dynamic pressure.

One other key concept here is the **first law of thermodynamics**. The first law of thermodynamics is the conservation of energy and states that **energy cannot be created or destroyed**. Imagine a river flowing out of a lake. The stagnant water in the lake has a dynamic pressure of zero, because it's velocity is zero. We'll call this state 1. The water at the mouth of the river does have some velocity, thus it has some dynamic pressure. We'll call this state 2. Because there is no energy being added to the fluid between state 1 and state 2, and because pressure can be thought of as the amount of energy per unit volume in a fluid, you should agree that the total pressure between states 1 and 2 should be equal. *Thus*, you should also agree that this means that the static pressure at state 1 is higher than the static pressure at state 2. ***Thus***, you should understand that as you increase the velocity of a fluid, the static pressure decreases and the dynamic pressure increases while the total pressure remains constant.

static pressure + dynamic pressure = total pressure

$$p_s + \frac{\rho V^2}{2} = p_t$$

$$\left(p_s + \frac{\rho V^2}{2}\right)_1 = \left(p_s + \frac{\rho V^2}{2}\right)_2$$

$$p_1 - p_2 = \frac{1}{2}\rho(V_2^2 - V_1^2)$$

and $A_1V_1 = A_2V_2$

Therefore,

$$A_2 < A_1, \quad V_2 > V_1$$

$$V_2 > V_1, \quad p_2 < p_1$$

decreasing area = increasing velocity

increasing velocity = decreasing pressure

Figure 7: Relationship between velocity, pressure, and cross sectional area characterized by the Bernoulli equation.

Boundary Layer

As a fluid flows over a surface, friction between the surface and the fluid near the surface causes the fluid to slow down. It can almost always be assumed that the fluid velocity at the surface is zero. A certain distance away from the surface, the fluid velocity is equal to the *free stream* velocity. The region between the surface and where the fluid velocity is less than the free stream velocity is called the **boundary layer**. For most fluids (Newtonian fluids), you'll see that fluid velocity has a parabolic relationship with distance from surface, as can be seen in figure 8.

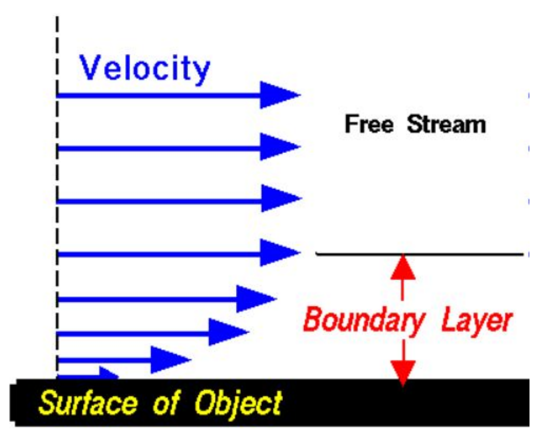


Figure 8: Velocity of a fluid in and near boundary layer

As a fluid gets slowed down by more and more friction, the boundary layer grows and its thickness increases (shown in figure 9). This is referred to as the boundary layer *developing*. The characteristics of a boundary layer are dependent on fluid properties like fluid density and viscosity, flow characteristics such as free stream velocity, and the micro-(surface roughness) and macro-(contour) geometry of the surface.

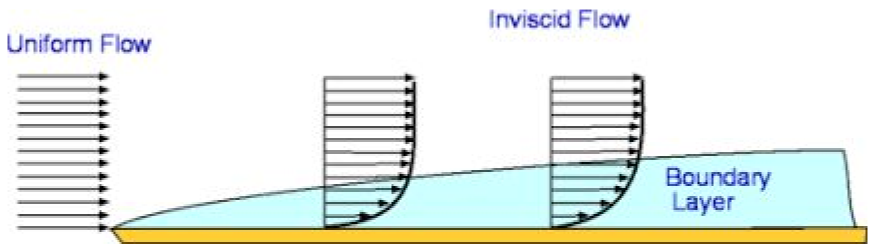


Figure 9: Development of a boundary layer on a flat plate.

As this boundary layer develops, the increasing thickness of the boundary layer creates an adverse pressure gradient. This means that as the boundary layer thickens, the static pressure in the boundary layer increases. This adverse pressure gradient, along with the surface friction causes the overall boundary layer velocity to decrease. If this adverse pressure gradient is too high, it can actually reverse the direction of flow within the boundary layer (shown in figure 10). This reversal of the boundary layer is referred to as flow separation. We'll talk more about flow separation after I introduce airfoils.

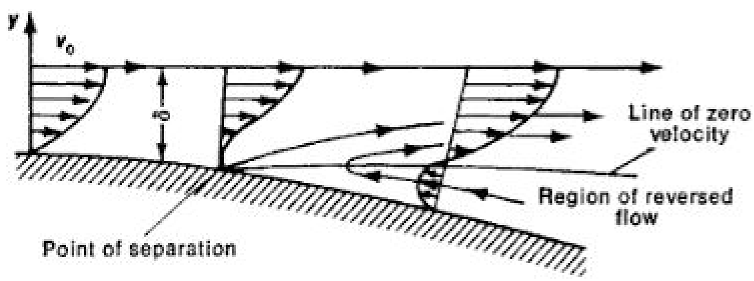


Figure 10: Reversal of boundary layer due to an adverse pressure gradient causes flow to separate from surface.

Aerodynamic Devices

Airfoils

Airfoils are just fancy shapes that generate downforce without generating much drag. If you try googling how exactly airfoils work, you'll get a lot of confusing, conflicting results, and while the answer to this question might be important to know for design finals, it is not critical to understanding how to design an effective FSAE aero package. The most important thing to understand about how an airfoil works is that its fancy geometry causes air to move fast on one side and slow on the other, thus generating low static pressure on one side, and high static pressure on the other.

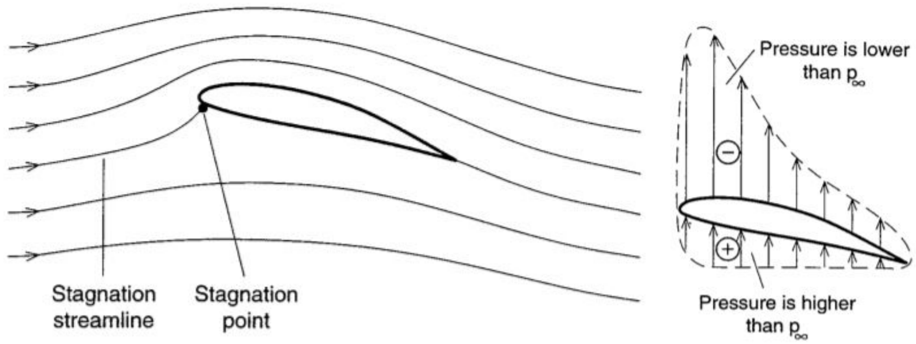


Figure 11: Streamlines near an airfoil (left), and the resulting pressure distribution (right) [1, p. 101]. Note: many figures in this report show airfoils used to generate lift. On the racecar, we use inverted airfoils, used to generate downforce. Keep in mind which side is the low pressure side and which is the high pressure side when looking at these figures.

The performance of an airfoil can be evaluated by observing the coefficient of lift at different air velocities and at different angles of attack (AOA). If either the angle of attack or the freestream air velocity is too high, stall can occur. Stall occurs when air separates from the low pressure surface of the airfoil. This flow separation causes the air velocity to decrease, increasing static pressure, resulting in a sharp decrease in lift and increase in drag.

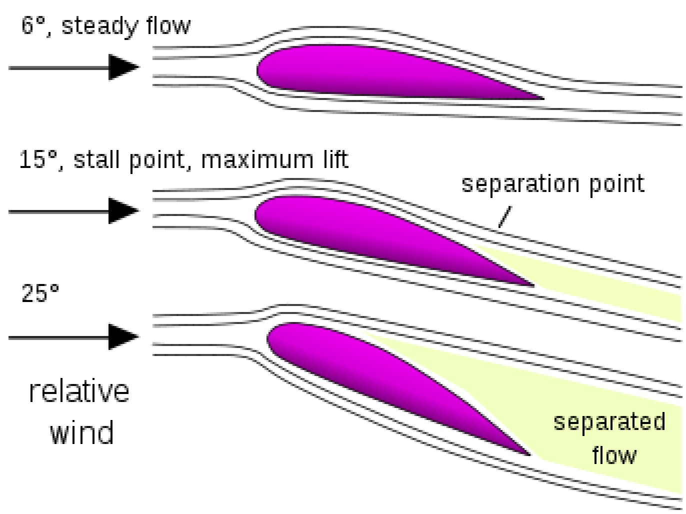


Figure 12: Flow tends to separate off the low pressure surface of an airfoil as AOA increases.

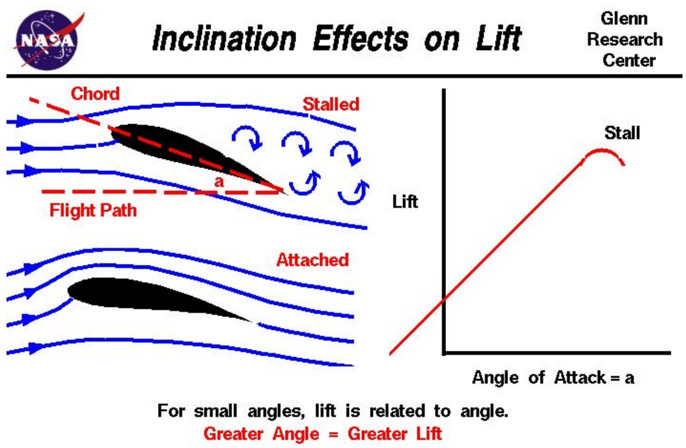


Figure 13: Relationship between lift and AOA. Stall occurs when AOA is increased past peak lift.

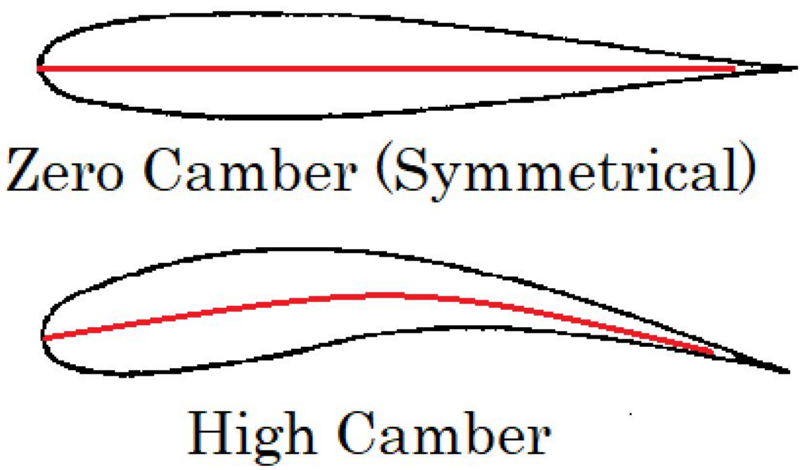


Figure 14: Airfoil camber.

Stall can also occur due to the **camber** of an airfoil (see figure 14). As the radius of curvature of the low pressure surface decreases (camber increases), it is easier for the momentum of the fluid to carry itself off of the surface. I often use my "ball rolling down a hill" analogy© shown in figure 15. The radius of curvature on the bump in the left sketch is larger than the one on the right. The momentum of the ball lifts the ball off the ground in the right sketch, but not in the left. The momentum of the ball is dependent on the ball's mass and velocity. This analogy still holds with a fluid. Imagine a fluid particle "rolling" along the surface of an airfoil. The particle's momentum (defined by its velocity and density) can launch it off of the surface if the radius of curvature is too small. High camber airfoils are considered more aggressive and can generally produce more lift at the same AOA and air velocity as an airfoil with less camber.

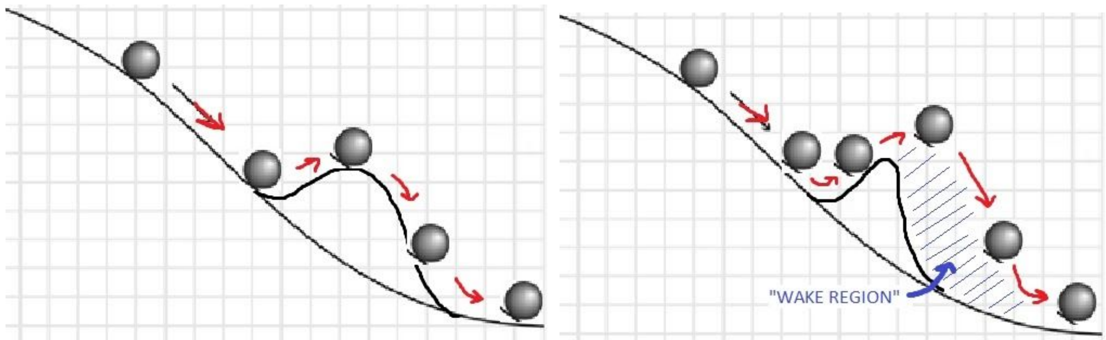


Figure 15: Flow separation due to high negative camber is analogous to a ball catching air when rolling over a sharp bump.

As described in the boundary layer section, flow separation occurs when boundary layer flow reverses. This can be observed in CFD simulations as shown in figure 16.

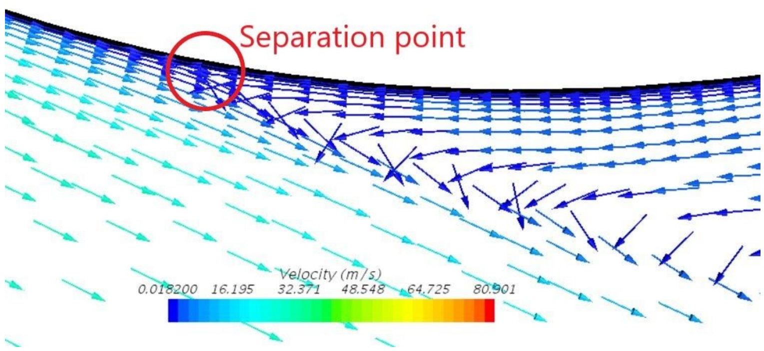


Figure 16: Flow separation observed in CFD.

One way to delay separation is with the addition of a **Gurney flap** (also known as a **wicker**). Gurney flaps create vortices at the trailing edge of the airfoil, increasing the airfoil's effective camber (making it effectively less negative), as shown in figure 17. These are useful in delaying separation without needing to adjust the geometry or position of the airfoil itself.

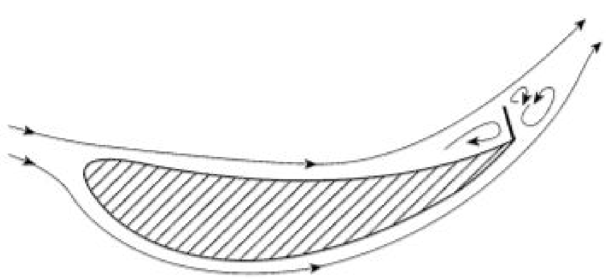


Figure 17: Effects of adding a Gurney flap

Multi-Element Airfoils

Multi-element airfoils can have higher effective camber than an equivalent single-element airfoil before experiencing flow separation. Figure 18 helps explain what I mean when I say "equivalent" in regards to single- vs. multi- element airfoils.

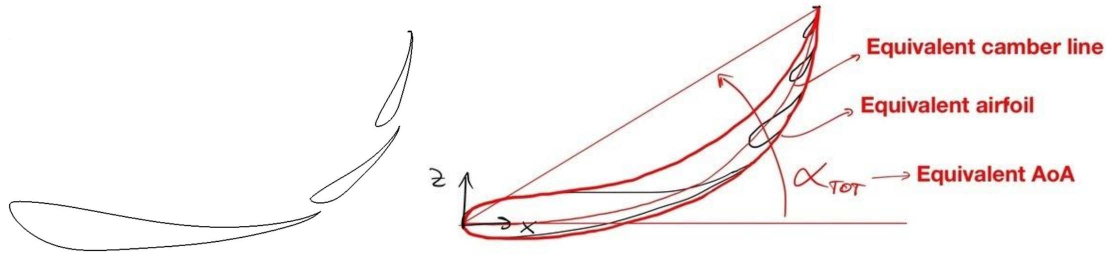


Figure 18: Equivalent airfoils, single-element vs. multi-element

Multi-element airfoils are one of the best ways to maximize downforce or lift given a limited amount of space. There are a few reasons why a multi-element airfoil can outperform a similar single-element airfoil and they are explained in depth in [2, Secs. 5.3-5.9]. A few key reasons that are easiest to understand are as follows:

1. As discussed in the boundary layer section, as a fluid stream interacts with a surface, the boundary layer requires some distance to fully develop. In a single element airfoil the boundary layer can increase in thickness along the length of the chord, as shown in figure 19.

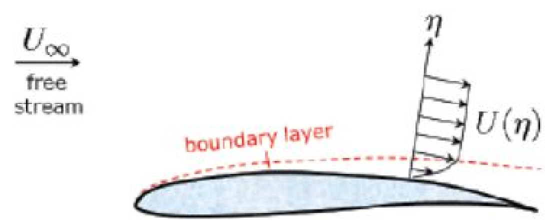


Figure 19: Development of boundary layer along chord length of single-element airfoil.

However, with a multi-element airfoil, because the geometry is broken up by each element, this boundary layer "resets" at the leading edge of each element, seen in figure 20. This causes the boundary layers of multi-element airfoils to be thinner overall than the boundary layer on a single-element airfoil. **A thin boundary layer will have less of a tendency to separate than a thick boundary layer.** Therefore preventing the multi-element airfoil from stalling before an equivalent single-element airfoil.

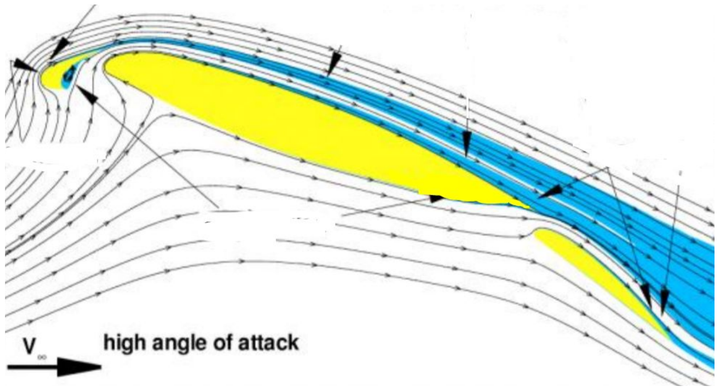


Figure 20: Development of boundary layer along chord length along pieces of multi-element airfoil.

2. As the boundary layer develops, friction dissipates energy from the flow causing the flow velocity to decrease. This increases static pressure of the boundary layer and increases the tendency of the boundary layer to separate from the surface. In multi-element airfoils, the interface between the trailing edge of one element and the leading edge of the boundary layer acts as a nozzle, increasing fluid velocity at each element leading edge. This can be seen in figure 21. This increase in velocity decreases static pressure in the boundary layer, decreasing separation tendency. This further contributes to the improved performance of multi-element airfoils.

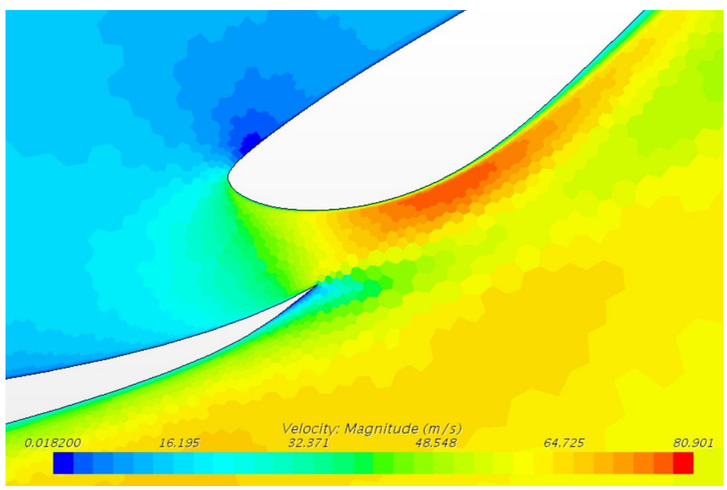


Figure 21: Flow accelerates as it transitions from the trailing edge of one element to a leading edge of another.

Underbody Geometry and Ground Effect

DISCLAIMER: I don't fully understand the mechanics behind ground effect so take this section with a grain of salt. If someone can explain this better than me, feel free to update this section.

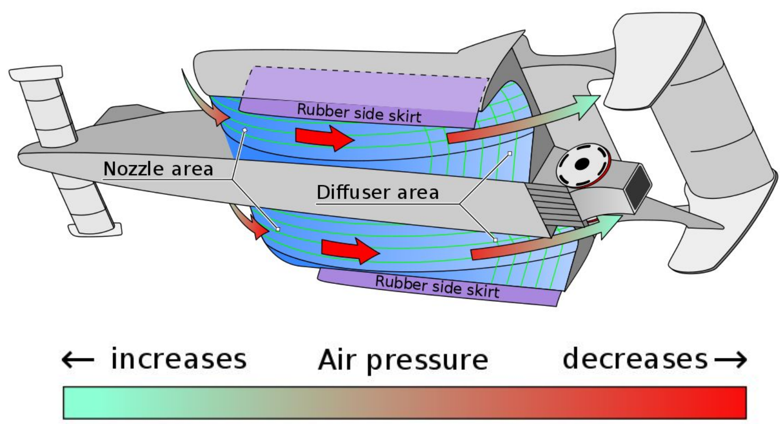


Figure 22: The effect of underbody tunnels or venturis on underbody pressure

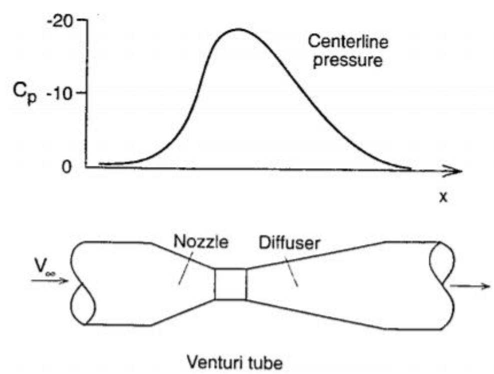


Figure 23: Bernoulli's relationships observed in a converging-diverging nozzle (venturi).

Bernoulli's equation lets us understand that in a venturi, (as shown in figure 23) the highest velocity and lowest static pressure will occur at the smallest cross sectional area. This concept is used in underbody geometry to create suction underneath the car. This is especially beneficial for race cars as the ground is stationary while the car moves over it. In reality, the road surface underneath the car does not have a boundary layer. Comparing the left side of figure 24 (boundary layer on road surface) to the right side (no boundary layer on road surface), the air velocity under the car will be lower on the left than the right. Therefore, the non-existence of a boundary layer on the road surface creates low static pressure zones underneath the car. This phenomenon is called "ground effect". Ground effect essentially generates extra downforce, while producing minimal drag, or even decreasing drag.

I'm not confident that this is the only mechanism that contributes to ground effect, or if it is even the primary mechanism. Someone should do some more research and update this section.

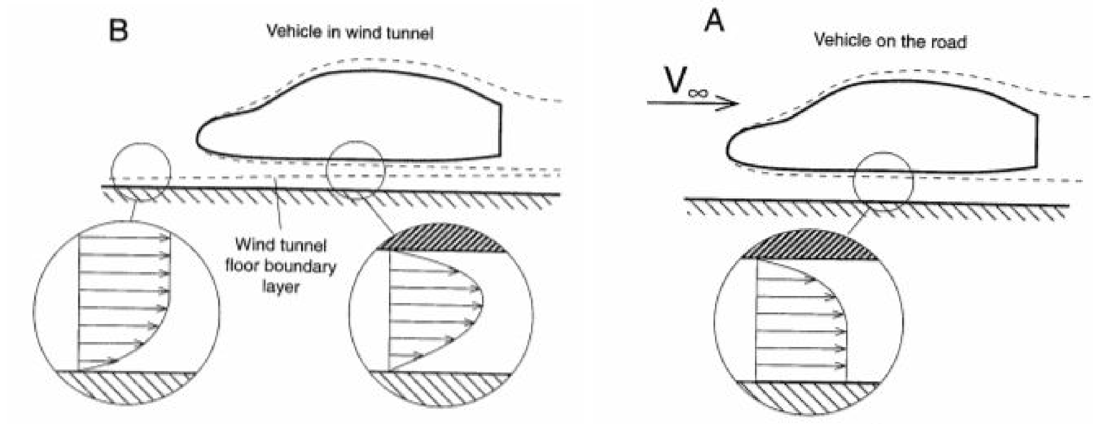


Figure 24: Road surface boundary layer in wind tunnel (left) vs. on track (right) [1, p. 73]

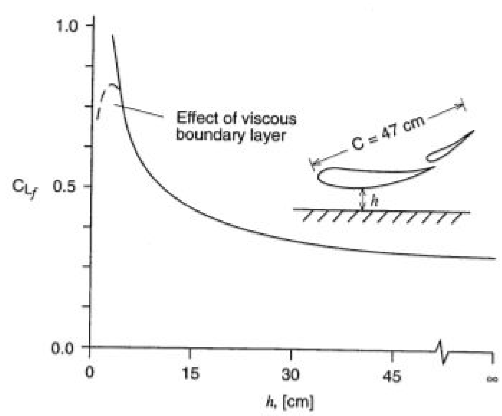


Figure 25: Effect of ground proximity on downforce produced by airfoil [1, p. 170].

As shown in figure 22, side skirts are used to seal the low pressure air inside the tunnel and prevent relatively high pressure ambient air from making its way underneath the car. They also prevent turbulent, high static pressure air coming off of the tires and going under the car. The effectiveness of these side skirts change drastically during the course of a lap as the size of the gap between the sideskirt and the asphalt changes with ride height, pitch, and roll. However, the Lotus Type 78 used brushed or rubber side skirts that allowed for a dynamic seal. This innovation was a major factor in Team Lotus's Constructors' World Championship and Mario Andretti's Drivers' World Championship victories in the 1978 F1 season.

Figure 26: Mario Andretti racing the Lotus Type 78 in the 1977 US Grand Prix West at Long Beach, CA.

While these sliding side skirts are proven to be very effective, unfortunately dynamic or non-rigid side skirts are not allowed by FSAE rules. Unsprung side aero is allowed however, and would allow for the gap between side aero and the asphalt to be independent of suspension travel but we will discuss unsprung aero later in this report.

Another innovative method to improve effectiveness of underbody aero is what is referred to as a “blown diffuser”. The Brabham BT46 (known as the “fan car”) was a car developed by the Brabham F1 Team for the 1978 F1 season as a response to the Lotus Type 78. The Brabham BT46 used a massive fan mounted at the underbody diffuser outlet “for cooling purposes” and forced a massive amount of suction underneath the car. The car is not known to be well developed outside of this innovation, however the car did go on to win the Swedish Grand Prix that season with Niki Lauda behind the wheel. The fan car was banned the following season.

Figure 27: The 1979 Brabham BT46B: “The Fan Car”.

The idea of a blown diffuser is technically not allowed in FSAE (however the rule prohibiting blown diffusers was removed from the FS Germany 2020 ruleset), so the “designed exclusively for cooling” clause of rule T.7.2.2 causes some controversy (figure 28) . UM-Twin Cities has a large diffuser with radiators mounted on the tunnel outlets and a large fan strapped onto each radiator core. This way they can say that the fans are for cooling purposes only. I am unsure of the performance gained by doing this, but a simple CFD study could be performed to find out. Mounting radiators on the diffuser outlet would require a lot of thought towards mounting, plumbing, and heat management (specifically for the c car), as well as protecting the radiator cores from rocks, cones, etc.

T.7.2.2 No power device may be used to move or remove air from under the vehicle except fans designed exclusively for cooling. Power ground effects are prohibited.

Figure 28: Rule T.7.2.2 taken from FSAE Rules 2020 V2.1, prohibiting blown diffusers.

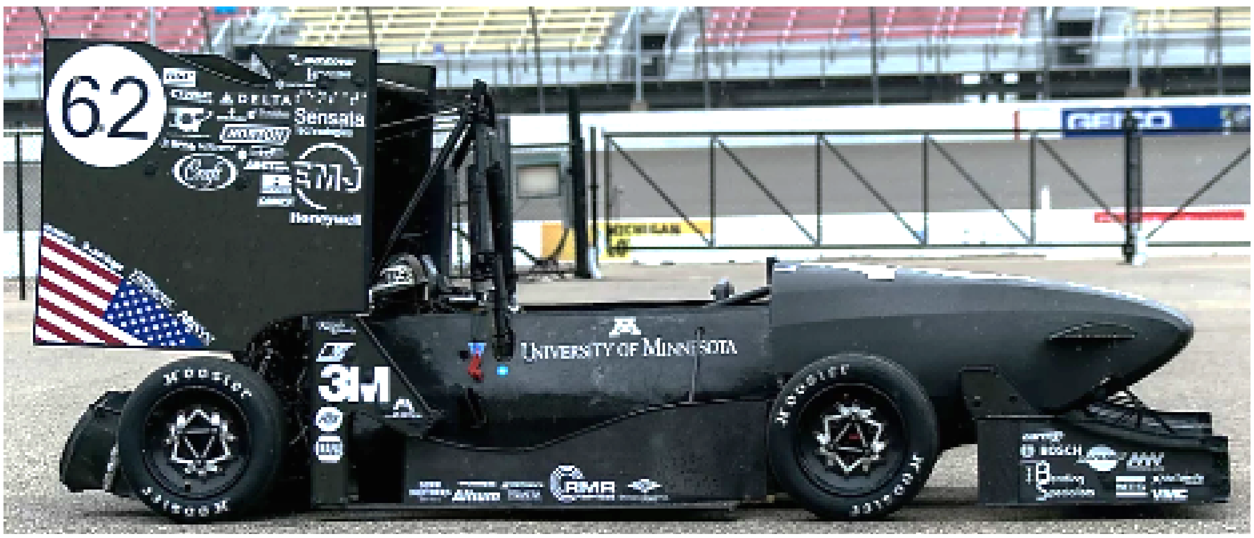


Figure 29: University of Minnesota - Gopher Motorsport's 2019 car. One of two rear fans can be seen at the bottom left of the photo.

Although figure 22 shows a venturi-style underbody tunnel with both converging and diverging sections, the converging section is not necessarily required to make underbody tunnels beneficial. Having a diffuser on the rear underbody of the vehicle allows for the air to gradually slow down and return to near ambient pressure. This is shown in figure 30.

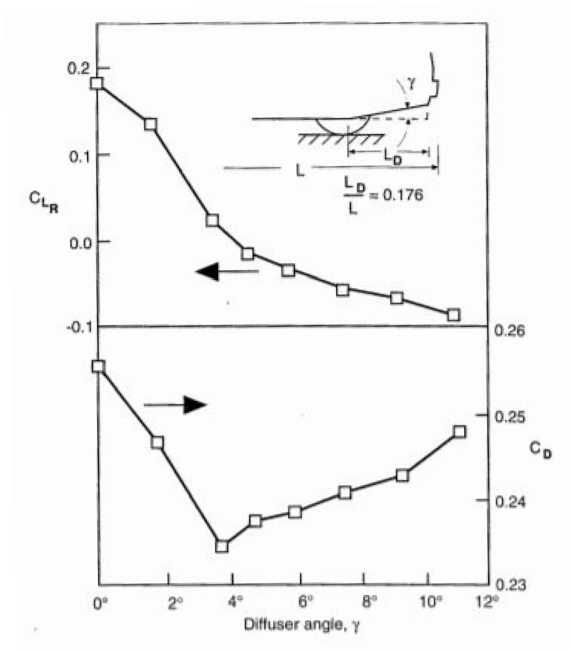


Figure 30: Effect of rear diffuser angle on drag and rear axle lift [1, p. 206]. (Note: C_{LR} = coefficient of lift in vehicle rear. Negative C_{LR} denotes downforce)

A diffuser along with a rear wing can be especially beneficial. A rear wing redirects air upwards, creating an updraft. A rear wing can decrease pressure at the diffuser outlet, increasing air velocity under the vehicle. This effect of the rear wing can augment the effect of the diffuser and make it more efficient. This effect is shown in figure 31. Notice how the vehicle with a rear wing has lower pressure along the underbody than the vehicle with no rear wing. While this combo-effect is beneficial in Prototype and F1 cars, it is probably not as relevant for FSAE cars due to the position of the rear wing being relatively high in comparison to these other race series. This fact could be checked in a CFD study analyzing underbody pressure distribution with and without a rear wing.

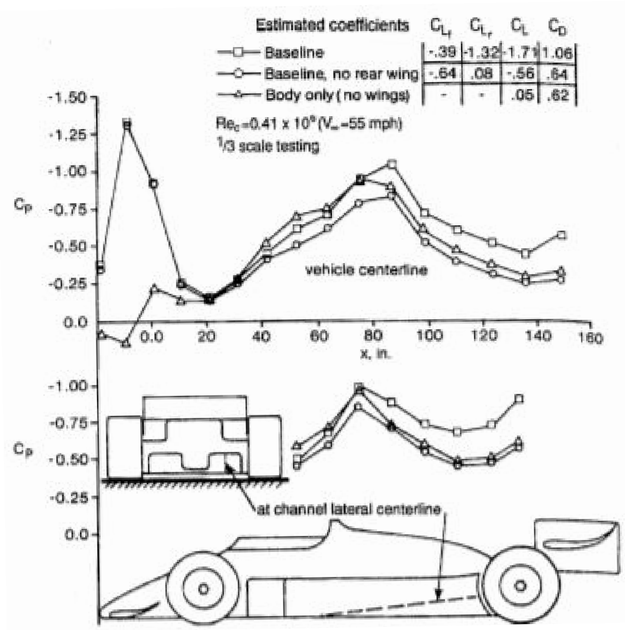


Figure 31: Effect of rear wing on an open-wheel race car's underbody pressure distribution along tunnel centerline [1, p. 208]

These diffusers are very prevalent with other FSAE teams, however. Undertrays have the potential to add a significant amount of downforce without adding much drag. A large benefit is that downforce is produced near the center of the car, meaning that the addition of underbody tunnels would not require any major changes to be made to the front or rear wing to maintain the same aero balance. Underbody aero does add weight and most importantly, it adds another huge manufacturing task for the composite team.

Figure 32: TU Münich's 2018 car featuring large diffusers built into their sidepod floors.

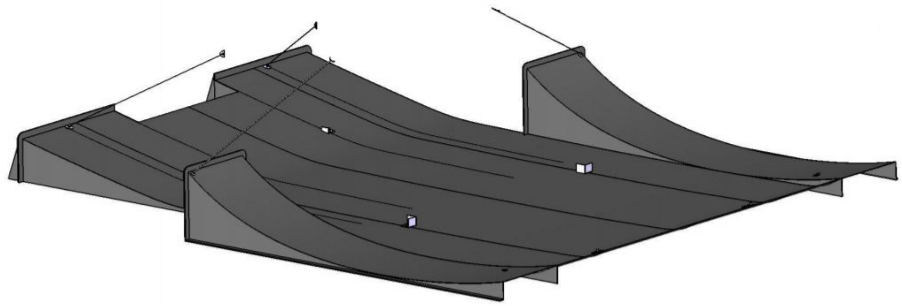


Figure 33: Western Washington University's Viking 61 diffuser design

Design Process

Disclaimer

I took responsibility for this project without having any knowledge or previous experience with race car aerodynamics. I kind of stumbled through figuring out this process throughout the year. In this section I document my process, and what I think I may have done incorrectly, or what should be different next time.

Background

The last redesign of the aero package was for the 217c. I believe this aero package used the same rear wing geometry as the 215 and 216, but the front wing was different. The 216 front wing had a massive anhedral main element and 2 tiers of flaps on each side. I don't quite understand how this car had such a large front wing without screwing up the aero balance, unless the wings attached to the sidepods or the underbody floor/diffuser were very effective.

Figure 34: WR-215 (left) and WR-216 (right) notice the huge anhedral front wings and side aero.

Figure 35: Render of WR-216 aero package. Note the underbody and side geometry. You can just see the diffuser peeking out the back.

Note the much smaller front wing design on the 217 in figure 36. This package redesign also removed the front wing end plate tunnels and multi-element side aero. The side wings underneath the sidepods only lasted for the 217. The following season had larger sidepods for new radiators and required flat plate floors instead of airfoils underneath the sidepods.

Figure 36: WR-217 at MIS. Note that the rear wing elements are trimmed down for the acceleration event.

Airfoil Selection

Wisconsin Racing's first aero packages on the WR-213 and WR-214 used the Wortmann FX 74-Cl5-140 MOD airfoil for its high coefficient of lift and its stable stall characteristics. The WR-215 swapped for the **Selig 1223RTL** airfoil due to its even higher coefficient of lift (figure 37). We have used this airfoil since. The Selig 1223RTL is denoted a low-speed airfoil because it achieves max lift at relatively low **Reynolds numbers**. It does produce high drag (its L/D is much lower than other low-speed airfoils), but in FSAE downforce is much more important than drag (I explain this in more detail in the next section). These factors, along with its relative ease of manufacturing, makes the Selig a popular airfoil among FSAE teams.

Past airfoil research has utilized airfoiltools.com. This free site has an incredible amount of data on hundreds of airfoils.

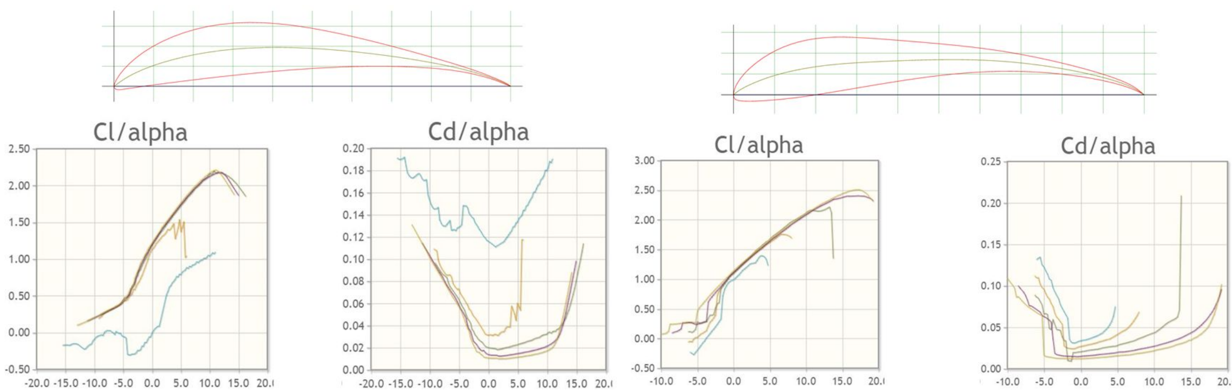


Figure 37: Wortmann FX 74-Cl5-140 MOD (left) versus Selig 1223RTL (right).

Architecture

I decided to keep the overall architecture of the 220 aero package similar to the 217/218/219 for a few reasons:

1. **Maintain well understood manufacturing processes.**

   An aero design is worthless unless it is manufacturable. Isaiah spent the 218 year designing a super fancy F1-style front wing that supposedly improved sidepod flow and downforce a crazy amount, only for it to be impossible to manufacture. I really wanted to avoid making the same mistake, especially because our composites team was very small and relatively inexperienced. They already had enough on their plate making two monocoques and two full aero packages. By keeping the same airfoil profile and chord lengths, existing molds could be used and we know how to manufacture the elements without destroying them. (Imagine trying to pull this (figure 38) skin off a mold, trying to fit a spar and end caps inside, and hoping it contributes to a sufficiently stiff wing.)

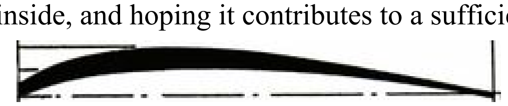
   

   Figure 38: Skinny airfoil. It's a bit of an exaggeration, but it gets my point across that selecting a new airfoil could have opened a whole other can of worms.

2. **Focus on understanding air management and wing/element positioning.**

   I already knew that it was possible to successfully argue the architectural design of the 217/218 aero package at competition, and starting this project off already knowing airfoil profile and size, and general structure of front and rear wing lightens the workload a lot. I wanted to focus this project on developing analysis tools and increasing the team's understanding of aerodynamic design/analysis overall. Once this knowledge base is solidified, it would become much easier to switch to make other architectural changes in future seasons. This could include changing airfoil profile, element size, number of elements, element twist/anhedral angle, etc.

Simulation

What is CFD?

Computational fluid dynamics (CFD) *is a science that, with the help of computers, produces quantitative predictions of fluid-flow phenomena based on the conservation laws (conservation of mass, momentum, and energy) governing fluid motion.* Basically, we can use computer software to predict the behavior of a fluid. You just need to tell the computer the properties of the fluid, the geometry that it is flowing through or around, and the fluid's **boundary conditions**. In our case, our boundary conditions are the vehicle speed (relative air velocity) and atmospheric pressure. These simulations can be very time consuming and computationally intensive. Thus, we are always trying to make the model as simple as possible, without sacrificing the level of desired accuracy.

Geometry

In CFD, all geometry is represented as a **mesh**. A mesh is just a finite amount of elements that represents a continuous body or volume. The fluid body itself is also a mesh, allowing for it to be represented as a finite number of cells. The computer can then solve for mass, momentum, and energy of the fluid in each cell.

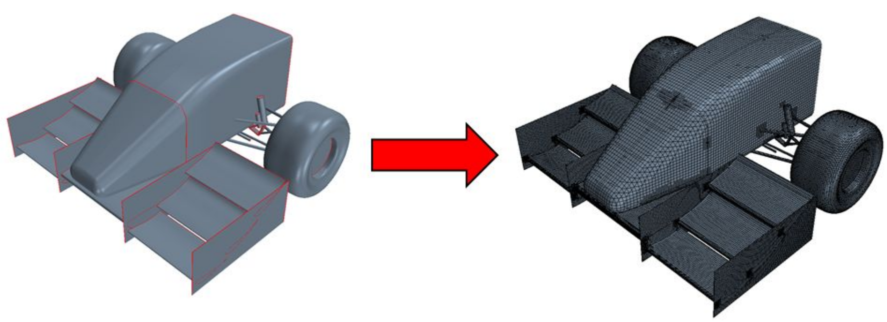


Figure 39: Continuous geometry (left) discretized into a mesh (right).

The size and shape of these cells can change the speed and accuracy of the model. Trimmed cell or hexahedral (cube-shaped) mesh is typically recommended for 3D external aerodynamic simulation. The smaller the cells, the more cells the computer must solve for, increasing computing time. Smaller cells also represent continuous geometry better than large cells. When selecting the cell size, there is a trade-off between computational expense and model accuracy. As cell size decreases, model output will change asymptotically. This asymptotic relationship seen in figure 41 (left) represents the diminishing returns associated with decreasing cell size. These plots in figure 41 are the result of a **mesh sensitivity** test. Mesh sensitivity describes how sensitive the output of the model is to mesh size. The results of these tests can be used to determine what size mesh should be used.

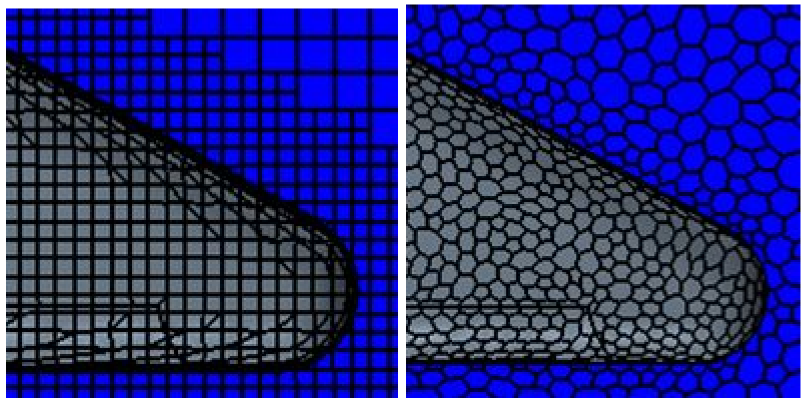


Figure 40: Hexahedral mesh (left) and polyhedral (right) of the nose cone (gray) and fluid body (blue)

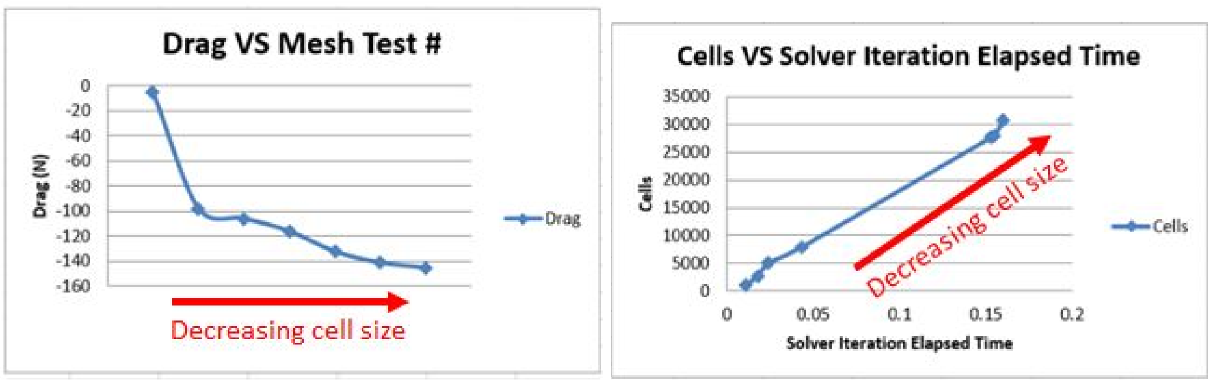


Figure 41: Model output vs. cell size (left). Model solve time vs. number of cells (right).

Performance Metrics

Imagine that you have three new rear wing designs that you want to compare to the original design. New Design 1 produces 25% more downforce but 13% more drag. Design 2 produces 8% more downforce and 13% less drag. Design 3 produces 8% less downforce and 8% less drag. It is nearly impossible to determine by intuition alone which design is best. It is important to be able to quantify exactly how much better one design is than another.

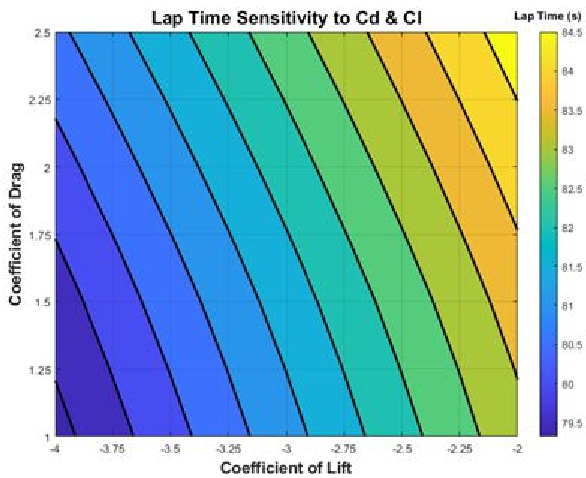


Figure 42: Lap time sensitivity to coefficient of drag and lift.

Using our lapsim model, I analyzed the sensitivity of drag and lift on lap times. The output of this simulation is seen in figure 42. You can see that the slope of these black lines is approximately -2. This tells us that if the aero package can generate units of more downforce while producing no more than 2 × units of more drag, lap times will decrease (i.e. lap time is approximately *twice* as sensitive to change in downforce as it is to change in drag). We can use this relationship to "score" different design iterations, with the highest score corresponding to the greatest performance gain.

New Designs 1, 2, and 3 are plotted over the lap time sensitivity plot in figure 43. Vectors drawn from the original design to each new design and understanding the "ideal" new design direction (perpendicular to the black lines) lets us create an equation to calculate a score for each design. Taking the dot product between each new design vector and the ideal vector tells us how close to ideal each new design is. This boils down to equation 3.

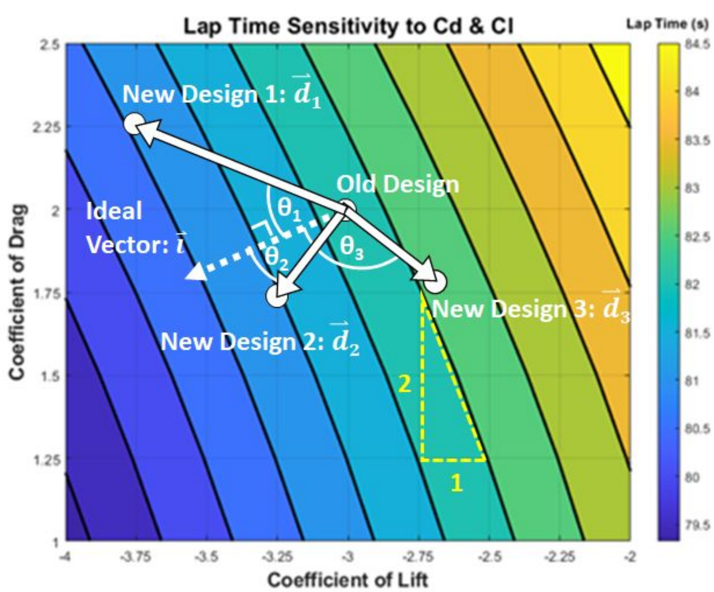


Figure 43: Lap time sensitivity with hypothetical designs overlaid.

$$\text{Score} = 2(\text{lift}_{new} - \text{lift}_{old}) - (\text{drag}_{new} - \text{drag}_{old})$$

Equation 3: Scoring new designs, derived from dot product between new design vector and ideal vector.

Model Types

Aerodynamic simulation took place in three stages: 2D, 3D (partial car), and 3D (full car). Each of these models had differing levels of **fidelity.** Fidelity refers to the degree to which a model or simulation reproduces the state and behaviour of a real world object or feature. An increase in fidelity typically corresponds to an increase in computational expense.

2-Dimensional

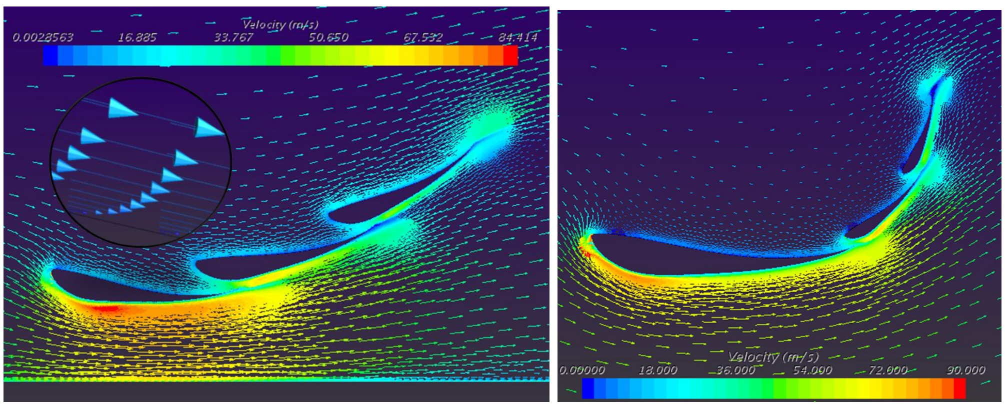


Figure 44: front wing (left) and rear wing (right) 2D CFD.

These 2D models were a fast way to find the best element positioning and spacing for the front and rear wings. They run ~20x faster per iteration than an equivalent 3D model, allowing for ~40 designs with differing element positions and gap spacings to be analyzed in one evening. Because each design took 1-2 minutes to simulate, it also provided a really good lesson on how gap size and overlap geometry affects downforce generation and flow separation. I also had very little experience with CFD leading up to this point, and starting off with building a 2D model was a great stepping stone towards mastering StarCCM+ (which I am still far from achieving). Vortices and end plate effects have key roles in wing performance and can not be captured in a 2D model. Because of this, these models are relatively low fidelity. In order to analyse the performance of the wings, the top scoring element positions were put into a 3D partial car model.

3-Dimensional (Partial Car)

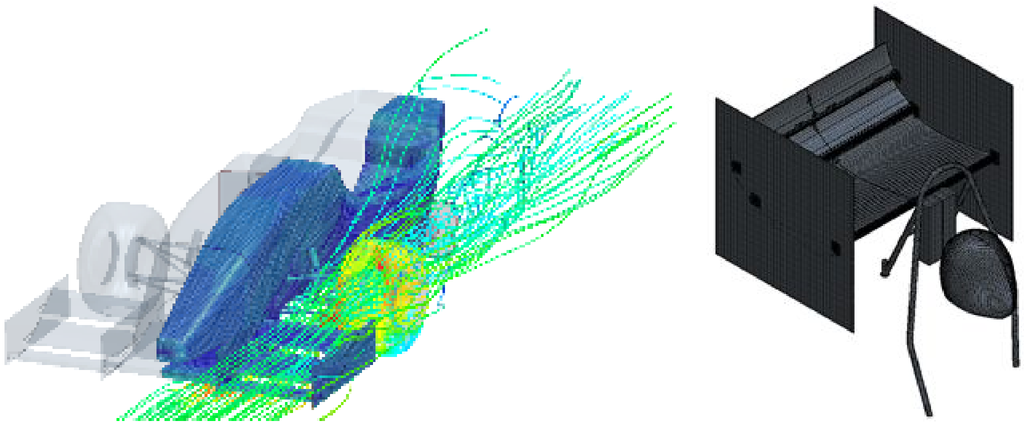


Figure 45: Front wing (left) and rear wing (right) 3D models

These partial car models allow for capture of vortices and endplate effects that can affect performance. Additional geometry such as the rear roll hoop, driver helmet, rotating tire, and monocoque can be modeled, letting us see how they affect airflow.

I first looked to see if the amount of flow separation in the 3D partial model was similar to the 2D model. If they differed, I continued tuning the element positions until I was happy with the result. For example: when analyzing the front wing, I had some instances of leading edge separation which contributed to terrible turbulence coming off the sides of the front wing (figure 46).

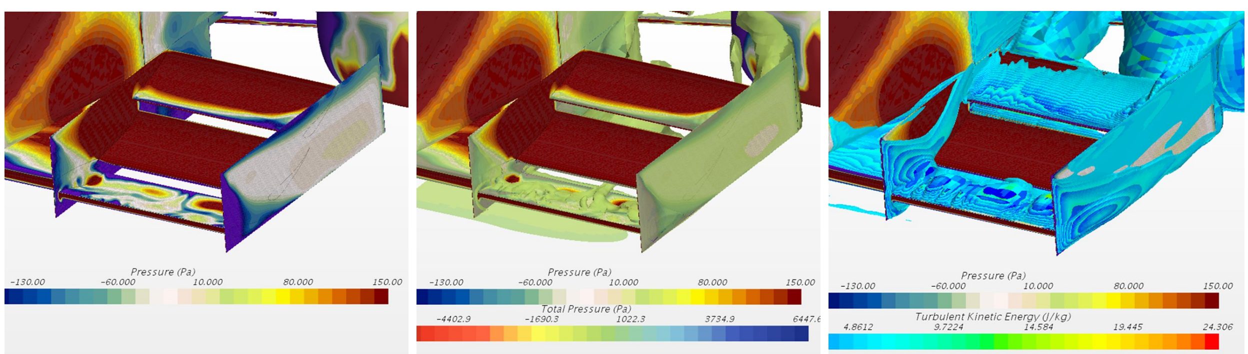


Figure 46: Pressure distribution on front wing (left), flow separation overlay (center), turbulence overlay (right).

This is an indication that the front wing is not producing as much downforce as it could. Besides producing downforce at the front of the car, the front wing also needs to supply clean air flow to the rear wing and sidepods. This extreme turbulence can affect performance of the rear wing and engine cooling system. Additionally, if this dirty air makes its way underneath the car, it will produce lift.

3-Dimensional (Full Car)

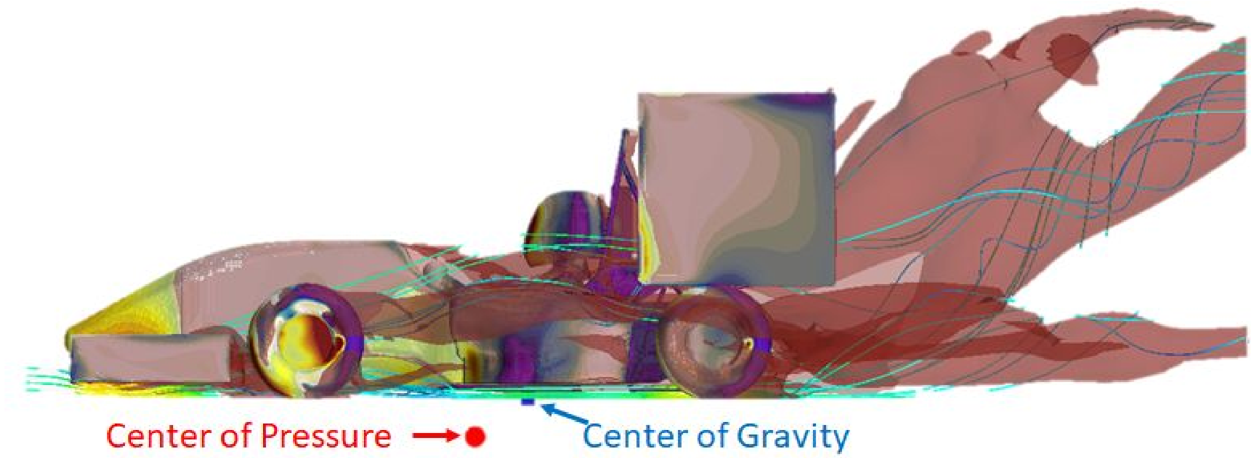


Figure 47: Full Car CFD - (combustion car CG location)

Once the front and rear partial models produced satisfactory results, the geometry parameters were entered into the full car model. Flow separation and surface pressure distribution can be analyzed in the full car model and compared to the partial car models. The rear wing is especially affected by the geometry of the front wing, wheels, driver, etc. which is not necessarily captured in the rear wing partial model.

One major factor analyzed in this full car model is the aero balance. The targeted location of the CP is just slightly rearward of the CG, and as seen in figure 47, the CP was initially too far forward. To push the CP backwards, the front wing was slightly detuned to produce less downforce

What I should have done differently

1. Element Position Optimization

StarCCM+ has an optimization tool which I tried to use to optimize wing element spacing. I parametrized the element position (X,Y and angle) and gap size, and set bounds for maximum wing height and length. The optimizer produced worse results than I did on my first try eyeballing each element position. This wasted 1.5-2 weeks of my time. I spent 6-7 hours one Friday evening iterating through about 40 different rear wing element position combinations manually, and ended up producing a pretty good result.

If someone could get the optimization tool to actually work, it would be super badass and the design judges would probably love it, but should only be a summer or new member project. I would recommend getting it to work in 2D, then again in a 3D model of the elements and end plates only, and then again with moving ground, moving wheel, coque, driver, etc.

2. CP Adjustments

Before detuning the front wing, I should have spent more time trying to improve the efficiency of the rear wing. I haven't put this into CFD, but I think adding a cowling on the backside of the driver headrest to help reconnect airflow before reaching the leading edge of the rear wing main element would have a noticeable effect on rear wing performance.

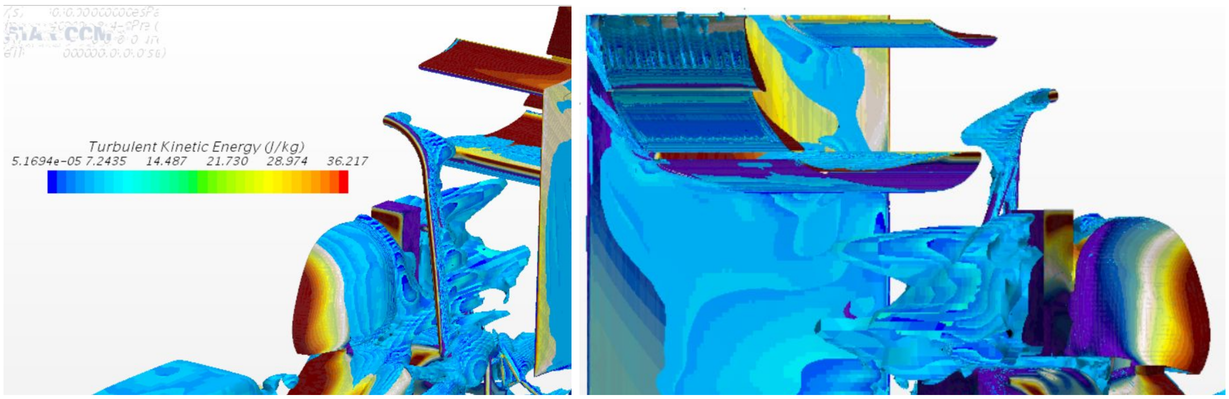


Figure 48: Turbulence caused by driver, headrest, roll hoop.

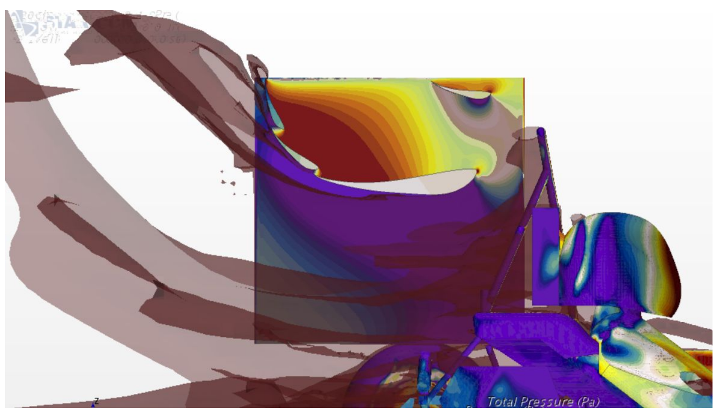


Figure 49: Flow separation caused by driver, headrest, roll hoop.

3. Planning for Mounting

I never gave mounting much thought during this process. Once I was designing end caps and had to place bolt holes, I discovered that almost every bolt head that mounts the front wing side elements to the center end plates interfered with the center element end cap (see figure 50 to see my wing anatomy naming convention). At this point, I had no more time left to continue iterating front wing element positioning in CFD. I ended up moving the entire front wing side assembly upwards so the bolts would clear. This essentially negated a lot of time I spent tediously adjusting the element spacing to achieve maximum performance. I adjusted the position of these elements accordingly in StarCCM+ just to make sure the CP was still rearward of the CG. By moving all side elements upwards, I was decreasing the amount of ground effect at the front wing. Intuitively, this should keep the CP in a stable position.

If I could go back and do this again, I would design the end cap truss structure so that the bolt heads do not interfere with the center end cap. This would have to be done so that the front wing can be trimmed and not interfere in any preset position.

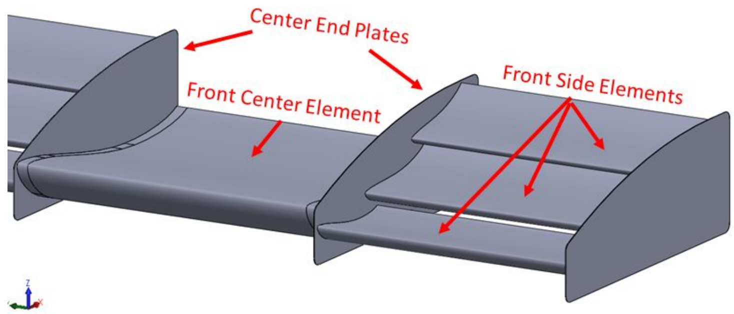


Figure 50: My naming convention for the front wing elements.

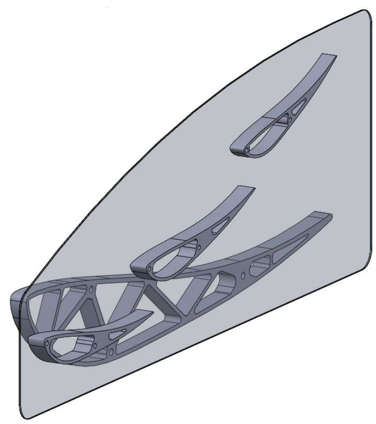


Figure 51: Interaction between side and center end caps on front wing.

4. Understanding the design space

In past years, the aero package was designed at an average cornering speed of 35 mph. I decided to favor driveability, and to maintain linear response from the car, I set my design point to 69 mph. My thought process was that if I designed the wings to have max performance at 35 mph, then the wings would stall at any speed faster than that. This could lead to unpredictability and poor driveability. The driver could have all the grip in the world at 35 mph, but nearly nothing at 40 mph. I still am unsure if this is the correct approach. This is an important question to answer before beginning the 221 season.

Validation and Testing

Purpose

Results from computer models are meaningless without any real life data showing that they are accurate. Additionally, it is tough to be confident in your design decisions unless you run some tests to prove that you actually achieved the performance that you predicted/desired. Therefore, in order to be confident in not only our analysis tools, but also the end result of those tools, we need to design and run some tests.

I don't know everything about correlating 3D aerodynamic models, but a great place to start is comparing simulated and actual coefficient of lift, coefficient of drag, sidepod inlet velocity, and flow characteristics such as turbulence, seperation, etc. By correlating a model, you are trying to answer the question: "does this model output reasonable results?" If you can argue that the answer is yes, then your model is correlated and therefore trustworthy.

Ideally, all of this data could be collected on-track because the environment that we design the car for is driving on a track. However, this environment is often hard to control (wind speed, air temp, road surface, etc.), and provides instrumentation limitations. These factors contribute to greater uncertainty in the data collected. Wind tunnel testing is a way to perform controlled, repeatable tests and allows for less instrumentation limitations. However it is obviously less representative of an on-track environment than an actual test track.

Wind Tunnel

Limitations

Wind Tunnel Size

In past seasons we have used the wind tunnel at Modine Manufacturing in Racine, WI. This wind tunnel is used for testing models much smaller than a FSAE car. According to Joseph Katz, a blockage ratio (frontal area of the vehicle divided by the cross-sectional area of the tunnel test section) of more than 14% is considered high and should be avoided [1, p. 89]. This is because upwash and outwash of air from the car can interact with the walls and ceiling of an undersized test section. This is not representative of how the air behaves on-track and therefore provides inaccurate results.

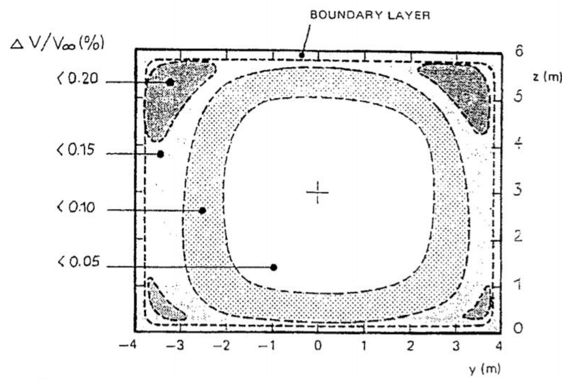


Figure 52: Variation in the test-section free-stream velocity in a wind tunnel. [1, p. 84]

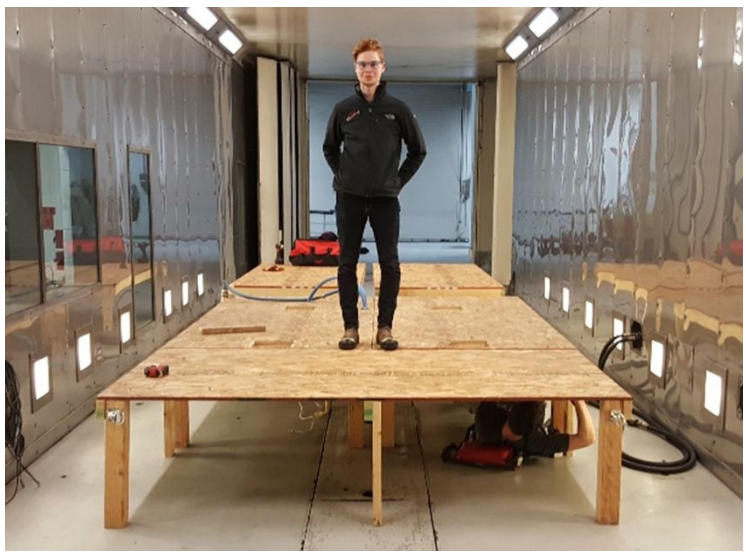


Figure 53: Modine wind tunnel. Dylan Vassar for scale.

I don't know the exact dimensions of the Modine wind tunnel, but you can easily tell from figure 53 that the blockage ratio will be much higher than 7%. While this isn't ideal, it also isn't the end of the world. The wind tunnel can still be used to correlate our CFD model if we include the wind tunnel geometry in our simulation. There are also research papers out there that develop correction factors for high block ratio wind tunnel models [3]. More work could be done with this to see if this is any use to our team.

Non-moving Road

The obvious difference between a wind tunnel environment and an on-track environment is in the wind tunnel there is no moving road surface and the tires are non-rotating. These have a significant impact on the aerodynamic performance of the car, however there are ways to simulate these factors in the wind tunnel.

As discussed in the ground effect section, ground effect is caused by the absence of a viscous boundary layer on the road surface. Unlike on a track, the air in a wind tunnel is moving in relation to the ground. This creates a boundary layer on the ground surface and this affects the performance of the aero package. The effect of this boundary layer must be minimized in order to collect accurate data in the wind tunnel. There are a few ways this can be done [1, p. 75].

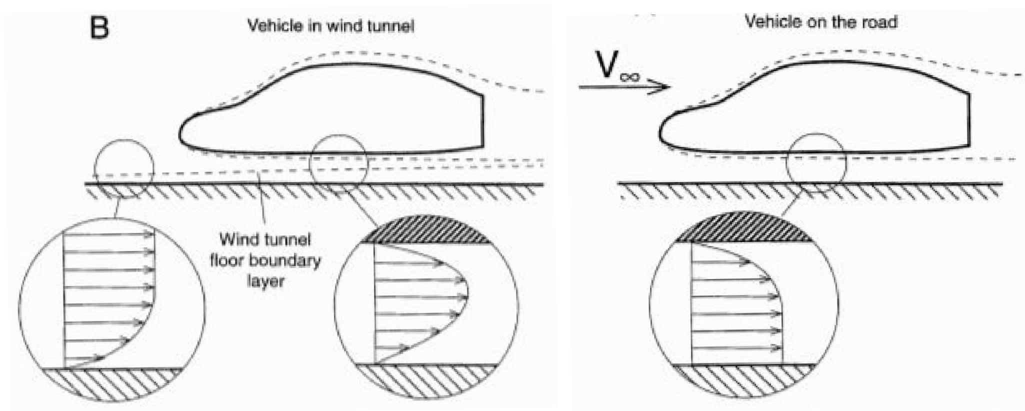


Figure 24: Road surface boundary layer in wind tunnel (left) vs. on track (right) [1, p. 73].

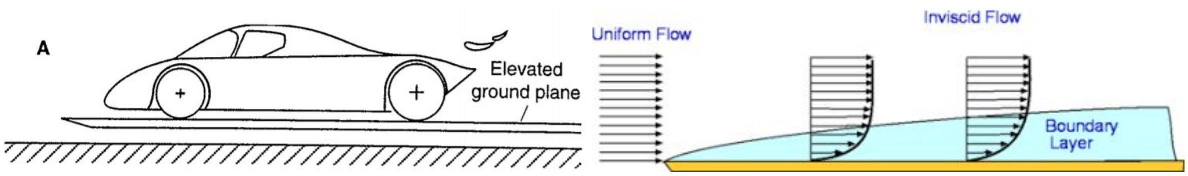


Figure 54: Method for simulating moving ground in a wind tunnel [1, p. 75] (left). Development of a boundary layer on a flat plate (right)

The simplest and cheapest way is to simulate a moving ground is to place the vehicle on an elevated surface (figure 54). As described in the boundary layer section, a boundary layer thickens as it develops. When the car is placed on an elevated surface, a "new" boundary layer is tripped close to the nose of the car, creating a much thinner boundary layer underneath the car than if the car were simply on the wind tunnel floor.

A rotating tire causes air to separate earlier and induces more turbulence than a non-rotating tire (figure 55). Air can be forced to separate in the same location as it would if the tire was spinning by strapping an angle iron to the top of the tire (figure 56). This also induces turbulence just as the rotating tire would (figure 57). This method is a little crude, but it works.

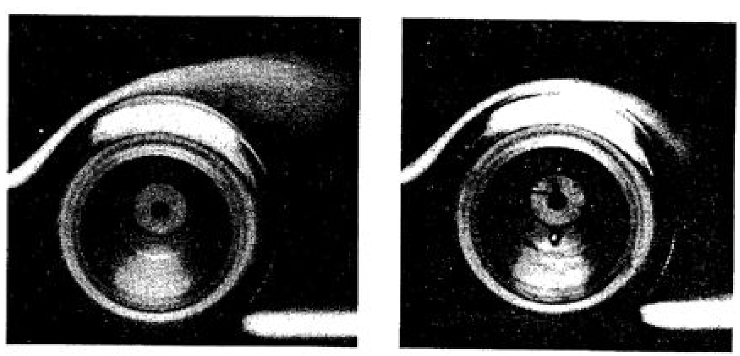


Figure 55: Visualization of the flow on rotating (left) vs. stationary (right) wheel [1, p. 195].

Figure 56: Simulating rotating wheel in wind tunnel with angle iron.

Figure 57: WR-216 in the Modine wind tunnel

Test Methods

Downforce

Downforce can be measured in the wind tunnel by placing scales under each tire, recessed in the elevated platform. From these recorded values total downforce as well as front-to-rear CP location can be determined.

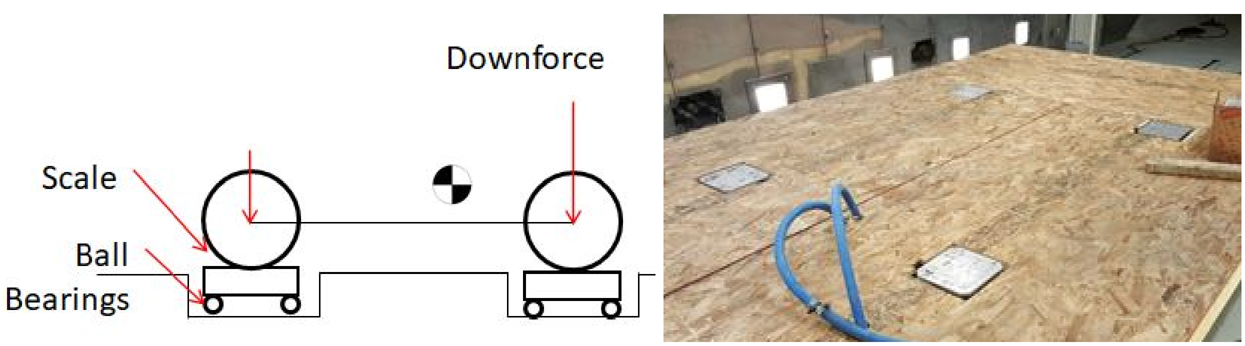


Figure 58: Wind tunnel scale setup

Drag

Drag is measured using 3 load cells mounted to the rear jack bar (figure 59). The position of these load cells must be measured in order to use their values to calculate any drag and lateral force. Because these load cells are rigidly mounted to the wooden platform, there is some downforce reacted through these load cells.

Figure 59: Wind tunnel load cell setup.

Flow Characteristics

Figure 60: Flow visualization with smoke wand (left) and tufts (right) in wind tunnel.

Tools like smoke wands or tufts (figure 60) allow for flow characteristics to be visualized. They are particularly useful for visualizing flow separation. If the flow separates off the underside of the rear wing in the wind tunnel but doesn't in CFD, then there's either a problem with the car, or with the model.

What To Do With The Data

Odds are the data collected in the wind tunnel will not be exactly what was output by the CFD model. An error of <10% is usually considered acceptable. If the error is greater than 10%, you'll have to figure out the source of that error. Unexpected flow separation can be an indicator that something is off in the model, whether there is some significant manufacturing imperfection not captured in the model geometry, or if the mesh size is too large, etc.

On-Track

Once the aero model is correlated with wind tunnel data, it is important to validate that expected performance is also seen on track.

Drag

Drag can be measured on track by performing a **coast-down test**. This test involves the driver getting up to a high speed, letting off the throttle/pulling the clutch, and coasting to a stop (if the track length allows for it). The total drag acting on the vehicle can be calculated with the equation shown in figure 61.

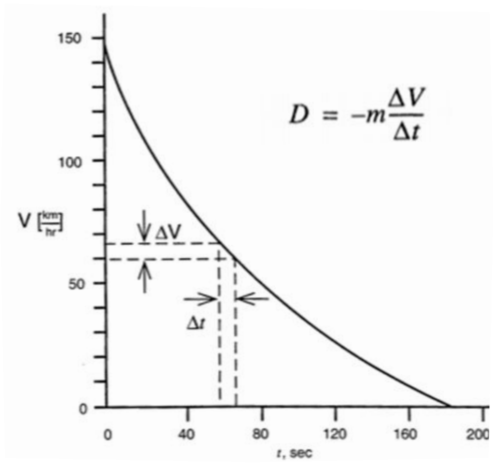


Figure 61: Typical variation of the vehicle's speed vs. time during a coast-down test [1, p. 59,60]

This drag also includes rolling resistance, friction, and rotational inertia. Overall bearing friction should be relatively negligible, but unless these factors are accounted for, the measured drag should be higher than the actual aerodynamic drag. Rolling resistance can probably be estimated from Calspan tire data, and rotational inertia is calculable. I'm not sure about the e car, but the inertia of the combustion powertrain and drivetrain is fully characterized (as of 218c). The aerodynamic drag should be relatively easy to back out of this total drag value.

A different method would not require any information regarding rolling resistance, friction, or rotational inertia. If the vehicle's powertrain is well characterized (thanks e car), and it is always known what the instantaneous torque at each wheel is, then the total force propelling the vehicle is known. This coupled with the vehicle mass and longitudinal acceleration can be used to calculate the force opposing the direction of motion (aero drag). See equation 4.

$$C_D \cdot A = \frac{T(n_{\text{engine}}) \cdot \frac{i_{\text{total}}}{r_{\text{rolling}}} - M \cdot G_{\text{long}} \cdot 9.81}{\frac{1}{2} \cdot \rho \cdot V^2}$$

Where: $T(n_{\text{engine}})$ = Engine torque as function of engine RPM
i_{total} = Total gear ratio
r_{rolling} = Tire rolling radius (of the driven wheels)
M = Total vehicle weight
G_{long} = Longitudinal acceleration
ρ = Air density
V = Vehicle speed

Equation 4: calculating drag on-track [4, p. 332].

Downforce

There are two main ways to measure downforce on track: measuring suspension travel, and measuring dynamic ride height. All downforce generated by the aero package travels through the suspension, through the tires, and is reacted by the road. If the stiffness of the springs is known, and the instantaneous displacement of the springs is recorded, then instantaneous downforce can be calculated. We can record instantaneous spring displacement via linear potentiometers across each shock (figure 62).

Figure 62: Shock potentiometer measures suspension travel.

The suspension members and the tires have some compliance/stiffness associated with them too. It is probably safe to assume that suspension members are rigid, but this assumption is not safe to make about the tires. Because of this, calculating downforce through the use of shock pots can be expected to have some small error associated with it. Although, the spring rate of the tires can probably be determined through Calspan tire data, and if the force through each spring is known, then the force through each tire is also known. Therefore, knowing the spring stiffness, tire stiffness, and spring displacement, it is possible to calculate the “displacement” of each tire.

Measuring instantaneous vehicle ride height is another way to measure downforce, but it still requires the knowledge of tire and spring stiffness. We have two laser ride height sensors, however they are very expensive and the model we have has a minimum ground distance of 6 inches or something. Some fancy rigging has to happen in order for these sensors to be implemented onto the car. Two sensors must be placed on the car (one in-plane of each axle) to measure aero balance.

For either method, some additional processing is required to account for longitudinal weight transfer. The vertical load at each axel due to longitudinal weight transfer during braking or when under power can be calculated given longitudinal acceleration and CG location. To avoid longitudinal weight transfer affecting the downforce measurement, some math will have to be done to account for this known weight transfer as a function of longitudinal acceleration. See equation 5.

$$L_F = MR_f \cdot SR_f \cdot (x_{\text{suspensionLF}} + x_{\text{suspensionRF}}) + \frac{M \cdot G_{\text{long}} \cdot 9.81 \cdot h_{\text{CoG}}}{WB}$$

$$L_R = MR_r \cdot SR_r \cdot (x_{\text{suspensionLR}} + x_{\text{suspensionRR}}) - \frac{M \cdot G_{\text{long}} \cdot 9.81 \cdot h_{\text{CoG}}}{WB}$$

With

MR_f = Front suspension motion ratio

MR_r = Rear suspension motion ratio

SR_f = Front suspension spring rate

SR_r = Rear suspension spring rate

$x_{\text{suspensionLF,RF,LR,RR}}$ = Suspension travel LF, RF, LR, RR (zeroed in static position)

h_{CoG} = Height of center of gravity from ground

Equation 5: Accounting for weight transfer and calculating lift at each axle on track [4, p. 332].

Aerodynamic Coefficients

To calculate lift and drag coefficients, instantaneous vehicle speed must be measured as well. Measuring vehicle speed via GPS will provide high uncertainty, and using wheel speed sensors will not account for wheel slip. Neither method accounts for wind either. Measuring vehicle speed should be done with a pitot tube. A pitot tube measures both static and dynamic pressure, which air velocity can be easily extracted from. A pitot tube can also record yaw angle and side wind velocity. To record accurate data, the pitot tube should be mounted high and forward where air flow disturbances are negligible. Once vehicle speed is known, and drag and lift force is known, then lift and drag coefficients can be calculated using equation 1 and equation 2.

Figure 63: Pitot tube on ETS's car. Some FSAE cars have permanent pitot tubes used for active aero control.

Flow Visualization

Visualising flow on track can be a trickier thing to do than in a wind tunnel, but it is still done. “Flow Viz” paint is often seen during Formula 1 tests. Basically thin coats of dyed oil are sprayed onto parts of interest on the car and it blows around and dries as the car drives. The colored streaks in figure 64 correspond to surface areas with high velocity airflow, while the clean area on the “Red” corresponds to low velocity flow, probably due to the presence of a wake.

Figure 64: Flow Viz paint used on a Toro Rosso F1 car.

The use of tufts (right - figure 60) can also be useful on track as well as in the wind tunnel. This is something that is often seen on the Baja vehicle. Attaching tufts and mounting a GoPro is another way to view the behavior of flow over the surface of the car. This method is a little more intrusive than using flow viz paint, but might be less messy.

In my honest opinion, unless the team is running out of tests to do, or if there is some uncertainty about how sensitive the wings are to flow separation in windy conditions, or with vehicle incidence angles not tested in the wind tunnel, on-track flow visualization is probably unnecessary. There are usually much better ways to spend time on the test track. But understanding options for testing flow characteristics on track are good to know.

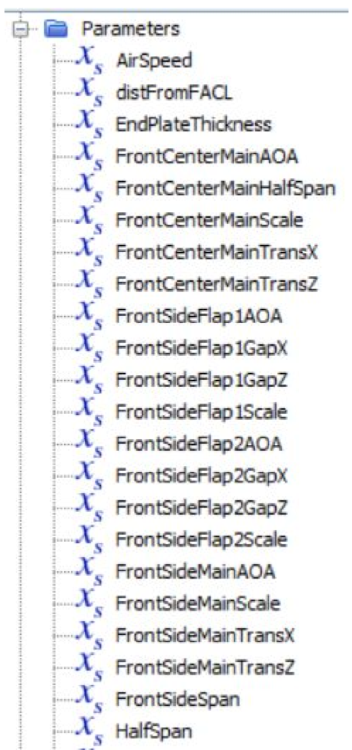
Appendix

Glossary

Reynolds Number: Reynolds number is an important dimensionless quantity used in fluid mechanics. A very low Reynolds number tells us that the flow is laminar, and a very high Reynolds number tells us that flow is turbulent. If you are unfamiliar with the concept of Reynolds number, for most of this report you can think of the Reynolds number being a dimensionless quantity that is proportional to velocity.

CFD Model

Multi-Element Parameterization



Physics Model

In a CFD model, you must define which methods and equations that the model will use to solve the state of each cell. In StarCCM+, this is called the **physics model**. For example: Are you analyzing 2D or 3D? Is the model transient or steady-state? Is your fluid incompressible? Etc. I'll describe the setting I selected for my model, but if you want to learn about when you would use other options, check out the [StarCCM+ documentation](). It is important to understand what each of these settings are and when they should be used. This is especially when troubleshooting why the model doesn't correlate to data, as well as when defending the validity/fidelity of your model.

- **Time - Steady:** This tells the computer that I will not be analyzing any transients. My boundary conditions will be constant.
- **Spacial Dimensions - 3D:** 3-DimensionalYou can choose between 2D or 3D. As I described, I did spend some time with 2D models, but a majority of my time was spent in the partial- and full-car models which were 3D
- **Equation of State - Constant Density:** My fluid is modeled as incompressible. This is usually a safe bet as long as all fluid flow is less than 0.3x the speed of sound (Mach 0.3)
- **Material - Gas:** my fluid is gaseous.
- **Gradients:** The gradients method discretizes governing differential equations in space and time into a system of linear algebraic equations. Computers are much better at solving linear algebra problems than differential equations.
- **Flow - Segregated:** Uses less memory, more robust and accurate for incompressible flow. Bad at capturing shock waves, high Mach flow.
- **All y+ Wall Treatment, Exact wall distance:** Affects boundary layer model by determining mesh geometry very close to surfaces
- **Viscous Regime - Turbulent:** In our case, flow turbulence cannot be neglected.
- **Turbulence Model**
  - **Reynolds-Averaged Navier-Stokes (RANS):** Time-averaged momentum equations used to solve fluid flow state. Making some assumptions allows for relatively simple RANS equations to be used to solve for flow turbulence.
  - **k-Omega:** The k-omega turbulence model is a type of eddy-viscosity-based model. Eddy-viscosity-based models assume that flow is either "statistically steady" or that there is significant turbulence due to flow separation and wakes. The "k" in k-omega refers to the turbulent kinetic energy, and the "omega" refers to the specific turbulence dissipation rate. This is an important model selection because without this turbulence model we may not get accurate predictions of lift and drag when flow separation occurs. This model is robust and accurate for various flow configurations (including separated flows, low-Reynolds number flows).

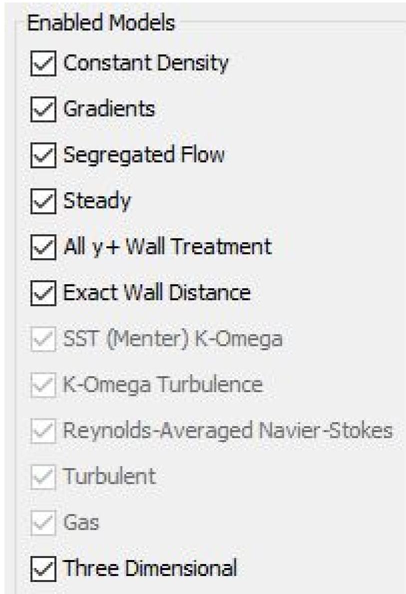


Figure 65: Selected physics models.

Still to Add

- 221 Recommendations
- Detailed CFD model documentation (I’ll probably put this in a separate document)
- Structural design process
  - Unsprung aero
- FEA (high level like I did in this doc with CFD)
- Detailed FEA model documentation (I’ll probably put this in a separate document)
- Active aero
  - DRS
  - Yaw-stability control
  - Is it worth it
- Reading Material
  - https://drive.google.com/file/d/1EtVkhcH2J4NHp1o5Ajc1ri9FYqn0KWsS/view?usp=sharing

References

[1] J. Katz, *Race Car Aerodynamics*, 1st ed. Bentley Publishers, 1995.

[2] A. M. O. Smith, “High-Lift Aerodynamics,” *J. Aircr.*, vol. 12, no. 6, pp. 501–530, Jun. 1975, doi: 10.2514/3.59830.

[3] D. Sahini and B. Tech, “Wind Tunnel Blockage Corrections: A Computational Study,” p. 136, 2004.

[4] J. Segers, “Chapter 13 Aerodynamics,” in *Analysis Techniques for Racecar Data Acquisition*, 2nd ed., SAE International, 2014.