The application of aerodynamics in the exterior design of racing cars

Yueheng Chen [1, †], Haoran Lin [2, *, †]

[1] Tsinglan School, Dongguan, Guangdong, 523000, China

[2] The Experimental High School Attached to Beijing Normal University, Beijing, 100032, China

[†]These authors contributed equally to this work.

* Corresponding author's e-mail: 101248@yzpc.edu.cn

Abstract. As a complex branch of mechanics, aerodynamics has been widely used in automobile design, especially in racing. In recent years, in the field of engineering application, the role of aerodynamics has become increasingly prominent, helping engineers solve many engineering problems that have not been solved before. At the same time, due to the development of aerodynamics, conditions have not been created for the rapid development of the automotive field. Therefore, this paper summarizes the basic principles of fluid mechanics, conducts experiments, and extends the laws from simple graphics to complex models to explain how aerodynamics is applied to the design of racing car shapes. The research shows that the rear fender and DRS system significantly improve the performance of modern vehicles; In the last century, ground effect and dolphin effect were both useful applications in automobile design, but they are no longer used. The development of modern technology has promoted the mechanism of racing design, and the latest development has yet to be determined.

Keywords: Aerodynamics, Racing cars, Basic principles, Modern vehicles.

1. Introduction

Fluid mechanics is widely used in industry, agriculture, transportation, astronomy, geology, biology, medicine, etc. Aerodynamics belongs to a branch of fluid mechanics, which is mainly used in aerospace technology, as well as in civil automobile design and racing design [1].

In the application of aircraft design, it is required to convert the force generated in the part affected by fluid mechanics into lift, and generate enough lift for the takeoff of the aircraft through a certain shape and angle. In addition, it is also required to calculate the takeoff speed of the aircraft to design a runway that is long enough. During the flight, it is also necessary to change the angle again to keep the aircraft in a balanced position in the air and adjust the appropriate angle to contact the ground when landing [2]. The application principle of civil cars and racing cars is basically the same, but the aerodynamics of cars is more to generate less force on the car through aerodynamics, so as to save the power output and fuel consumption of cars. When the car was just invented, the appearance design of the car was very bad, which would produce a lot of resistance [3]. Today, most of the car appearance designs are very similar because such a shape can produce as little resistance as possible and can meet the driving needs. The application on racing cars is just the opposite of that on airplanes. Racing cars need enough downforce to ensure stability and grip when passing corners. In some racing cars, the role of the huge tail fins is the same [4]. These huge tail fins are like an inverted airplane wing. They can provide a lot of downward force for racing cars to provide enough grip and stability.

Chen and Lin did two experiments to study some simple hydrodynamics [5]. They used hair dryers to provide lift for spheres of different sizes, and studied the height of spheres of different sizes being blown up and the stability of spheres in the air during the experiment. Another experiment is the paper airplane experiment, which studies the different flight trajectories of paper airplanes with different wing sizes folded from the same size paper during flight, and finds out the relationship between different trajectories and different wing sizes. These experiments provide simple and practical reference for understanding fluid mechanics.

The objective of this research is to understand the application of aerodynamics to car design, which is based on basic hydrodynamic principles.

2. Basic fluid dynamics

Discussing basic fluid dynamics principles helps the authors better understand the basis of car design and how to study further aerodynamic principles.

2.1. Hydrodynamic force & typical object shapes

When an object is immersed in a flowing fluid, due to the pressure and viscosity of the fluid, hydrodynamic forces are imposed on the object by flowing against and around it [6]. Depending on the shape of the object, the forces can be in any direction. Forces may also be unevenly distributed on the surface, resulting in torque causing an object to acquire angular acceleration. There are two main components of hydrodynamic forces: drag force and lift force.

2.1.1. Drag force

Drag force, F_D, is the net hydrodynamic force acting opposite to the relative motion of an object moving with respect to a surrounding fluid. It can exist between two fluid layers or between a fluid and a solid surface [7].

The drag force depends on several variables:

$$F_D = f(U, D, \mu, \rho, \varepsilon, \dots) \quad (1)$$

Where U is the magnitude of the flow velocity, D is the diameter of the object, ρ is the density, μ is the viscosity and ε is the roughness. This can be expressed as follows in terms of the drag force shown in Figure 1.

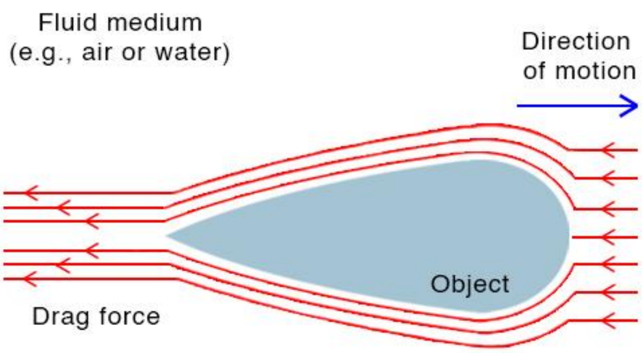


Figure 1. Drag force.

The drag force is often expressed in terms of the drag coefficient, defined as:

$$C_D = F_D \left(\frac{1}{2}\rho U^2 A\right)^{-1} \quad (2)$$

Where FD is the drag force, ρ is the density of the fluid, U is the speed of the object relative to the fluid, A is the cross-sectional area, and CD is the drag coefficient [8]. The drag force is usually calculated from

$$F_D = \frac{1}{2}\rho U^2 C_D \quad (3)$$

The drag coefficient CD depends on the Reynolds number Re and on the shape of the object. The Reynolds number is the ratio of inertial forces to viscous forces within a fluid with internal movement. It helps predict flow patterns in different fluid flow situations. Re is defined as

$$Re = \frac{\rho U L}{\mu} \tag{4}$$

Where ρ is the density of the fluid, U is the flow speed, L is a characteristic linear dimension, and μ is the dynamic viscosity of the fluid.

There is a direct relationship between drag coefficient CD and the Reynolds number Re [9]. For example, Figure 2 below shows drag coefficient CD for a sphere as a function of the Reynolds number Re. The dark line is for a sphere with a smooth surface (low viscosity), and the lighter line is for a sphere with a rougher surface.

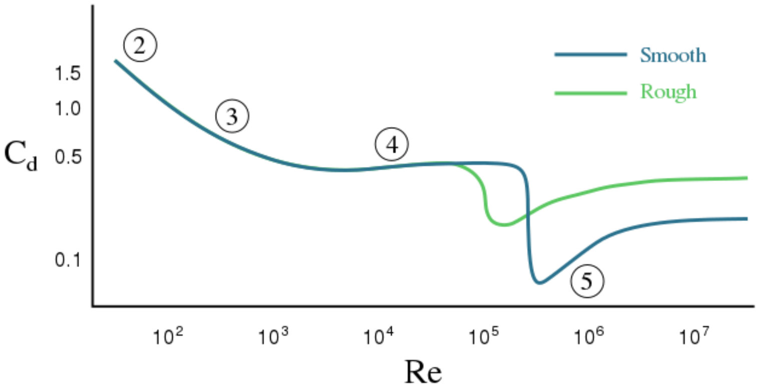
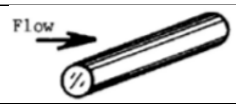
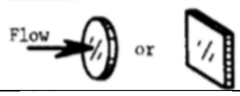
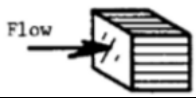
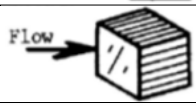
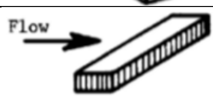
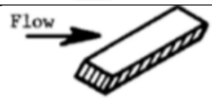
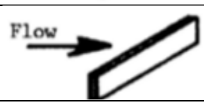


Figure 2. Drag coefficient of a sphere as a function of the Reynolds number.

Besides the effect of the Reynolds number, the drag coefficient CD also depends on the shape of the object [3]. As formula 2 above, the drag depends linearly on the size of the object moving through fluid. The frontal cross-sectional area of an object determines the form drag created by the pressure around the object. The drag can also be different with various direction of the same object [10].

Table 1. Drag coefficient for various object shapes [3].

Shape	Sketch	C_D
Right circular cylinder (long rod), side-on		1.20
Sphere		0.47
Rod, end-on		0.82
Disc, face-on		1.17
Cube, face-on		1.05
Cube, edge-on		0.80
Long rectangular member, face-on		2.05
Long rectangular member, edge-on		1.55
Narrow strip, face-on		1.98

It's found that the larger ratio of front area to thickness of an object, the smaller drag coefficient, because the viscous force is relatively smaller through a smaller area. And the drag coefficient is smaller when the flow fist touches an edge than perpendicular to a surface, as the cube in the table above [11]. Also, as Figure 3 shows, a smooth shape brings fewer turbulence and smaller drag coefficient than edges and corners. Turbulence, a chaotic flow, increases the viscosity of the flow. As a result, such structure with turbulence increases drag, which wastes extra energy for cars, planes, and other vehicles. To increase energy efficiency, a streamlined body is designed as below to create laminar flow and reduce turbulence, which is currently widely used in vehicle design [2].

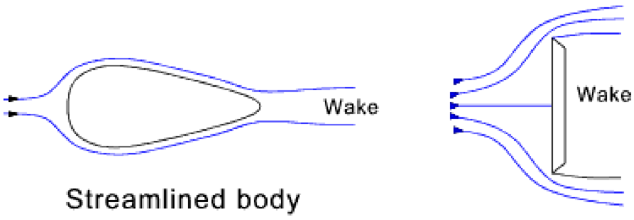


Figure 3. Laminar flow & turbulent flow [6].

2.1.2. Lift force

Lift force, F_L, is the component of total hydrodynamic force which is perpendicular to the direction of flow. It can exist between a fluid and a solid surface.

The lift force on an object depends on the asymmetry of the shape. The life force can be zero if the object is symmetric, and its direction can be either upward or downward relative to the direction of flow. Similar to drag coefficient, lift coefficient can be defined as

$$C_L = F_L\left(\frac{1}{2}\rho U^2 A\right)^{-1} \tag{5}$$

Where F_L is the lift force, ρ is the density of the fluid, U is the speed of the object relative to the fluid, A is the planform area of the object, and C_L is the lift coefficient.

As different shapes in Figure 4, airfoils as examples, a symmetric shape creates no lift force, and the more convex its surface, the larger the life force. And the direction of the lift force depends on the concavity of the surface. If the airfoil is convex up, the lift force is upward, just as real plane wings.

In aircrafts, lift coefficient can be influenced by angle of attack, as Figure 5 below. As angle of attack increases, lift coefficient increase. The air flow keeps parallel to the wing until angle of attack reaches stall angle. At that angle, the air begins to separate from the airfoil, so the lift coefficient begins to decrease.

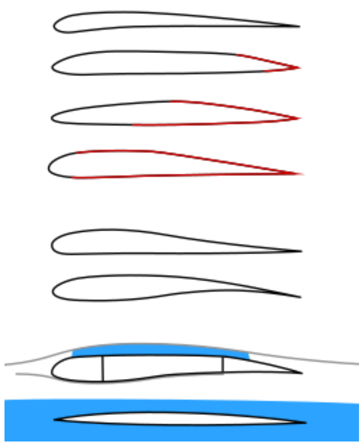


Figure 4. Airfoil shapes [6].

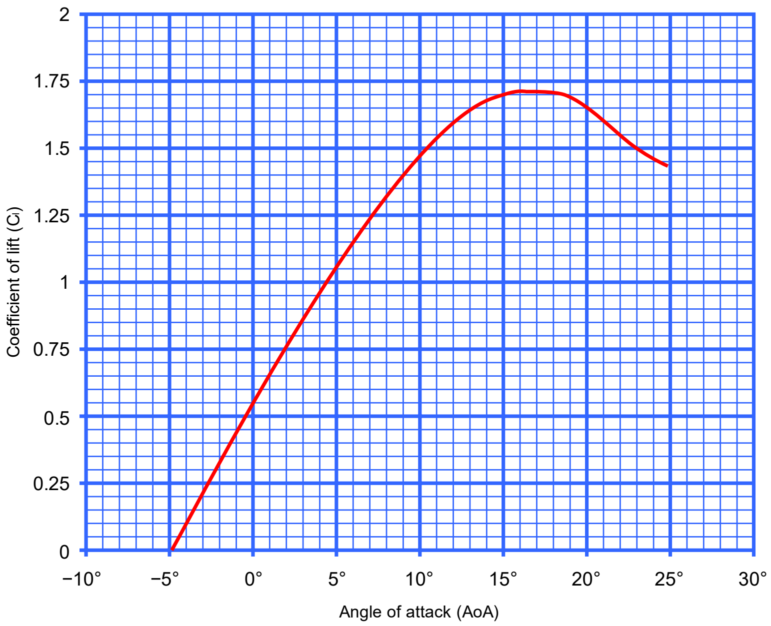


Figure 5. CL vs Angle of Attack [2].

2.2. Aircraft take-off principles and similar ones for racing cars

Essentially, Bernoulli equation is the basis of life force. Bernoulli equation is a formula simplified from the momentum conservation equation of fluid if the viscous forces are assumed to be negligible. Assuming that an incompressible flow is not viscous and that it does not change in time, there is

$$\rho g H + P + \frac{1}{2}\rho V^2 = constant \tag{6}$$

Where H is the height, ρ is the fluid density, P is pressure, g is the gravitational acceleration, and V is the fluid flow speed at a point on a streamline. This means that if height change can be negligible, pressure reduces as flow speed increases. Because in air flow, flow rate (volume of air flow per unit time) is constant, as in Figure 6, the air flows through longer distance on the upper surface per unit time. Then the air has higher velocity, with lower pressure. This difference of air pressure makes total lift force point upward.

In the process of aircraft take-off, the air plane needs to increase its lift force to overcome its gravity. As the plane speeds up, the lift force increases until it equals to the gravity. Then the plane can take off.

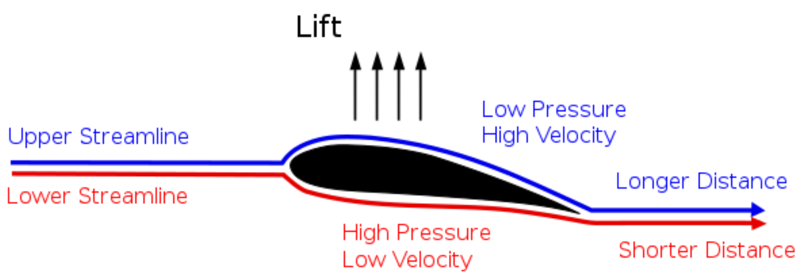


Figure 6. Bernoulli principle [9].

In car design, a car must cling to the ground instead of elevate as planes. Although a car can keep on the ground without any wing, the friction of the ground limits the max speed of the car, or the car would slip. This problem intensifies especially when the car is turning a corner. For instance, F1 racing cars need to grip the ground tighter so that they can turn at a higher speed. Engineers solve this problem by inversely installing similar structures like airfoils, so that lift force is added downward to increase friction proportionally. Then the F1 racers could increase their speed while stable.

2.3. Simple hydrodynamics experiments

2.3.1. Drag experiment

The first experiment introduces a ping pong ball levitating stably on a hair dryer. The general situation is that when a strong air stream is directed vertically upwards, a small object, such as a ping pong ball, can be suspended at a stationary position from the exit of the air stream. At such position, the drag force on the object is balanced by its gravity.

The goal of the lab is to measure and predict the position of the ball from the exit of the nozzle. If the measured position is close to the prediction, then the prediction can be approximately accurate. And if the deviation of the result is larger than 10%, the prediction is inaccurate.

Consider a turbulent jet emerging from a nozzle: the jet increases its range and reduces its speed as the distance from the nozzle is farther. And the flow has maximum speed at its central axis. As many studies have confirmed, the maximum speed, u, for a sphere can be defined as a function of the downstream distance along the jet, x.

$$u(x,0) = U_{max} = \frac{5Ud}{x} \tag{7}$$

Where *U* is the speed at the nozzle exit, and *d* is the diameter of the exit.

In the levitation balance, the ping pong ball has two forces, in equal amount and opposite direction: the drag force and the gravity. According to Drag equation,

$$Weight = F_D = \frac{1}{2} C_D \rho U^2 A = \frac{1}{2} C_D \rho \left(\frac{5Ud}{x}\right)^2 A \tag{8}$$

Then

$$x = \left[\frac{25}{2} C_D \rho U^2 d^2 A (Weight)^{-1}\right]^{\frac{1}{2}} \tag{9}$$

It is able to estimate the position of the ping pong ball using formula (9).

In the experiment, it is needed to measure and record drag coefficient CD, air density ρ, flux speed *U*, diameter of nozzle *d*, diameter of the ball *D*, and weight of the ball *W*. Then the value of *x* by calculation will be compared with the real height of the ball, to test the hypothesis.

Procedure:

1. To determine the flow rate from the dryer, measure the time it takes to fully inflate a trash bag with fixed volume. Then measure the dimensions of the bag to calculate its volume. The flow rate is Volume/Time.
2. To determine the drag coefficient CD, find appropriate CD in the graph of CD and Re.
3. In the experiment, first fix the dryer at a position, pointing upwards.
4. With the dryer on, place the sphere carefully suspended inside the jet, try to keep it levitating.
5. Perturb the ball a little (give it little kicks), to find out how stable its position is.
6. Take a video to measure the distance from the dryer’s exit to the levitated ball.
7. Measure the distance using software Tracker.
8. Repeat the same procedure for different balls, different dryer speeds.

Experiment results are as below:

Table 2. Ping pong ball results [1].

C_D	0.4
ρ	1.1kg/m^3
Volume of trash bag	0.0231m^3
Time needed to fill the bag	1.40s
U	10.4m/s
Diameter of the exit, d	4.5cm (0.045m)
Diameter of the ball, D	40mm (0.04m)
Weight of the ball	2.22g (0.028N)
Calculated position, x	0.054m
Experimental position in average	5.6cm

The results above meet the experiment data. The prediction is accurate enough, considering there exist unprofessional measurements done outside laboratory. In conclusion, it is possible to predict the position at which a small object levitates in a strong air stream.

Additional experiments:

In additional experiments, the air outlet of the hair dryer was an ellipse shape instead of a circle. However, four small balls of different sizes were compared. Because of this, the experiment was designed to get a general phenomenon rather than a concise calculation. Experimental procedure is similar to above, and the results are as Table 3.

Table 3. Additional ball experiments results [9].

Ping pong ball	
diameter	4cm
Maximum height from air outlet (center of circle)	9cm
Minimum height from air outlet (center of circle)	7.5cm
Average height from air outlet (center of circle)	8cm
Horizontal deviation	<1cm
Special	The ping pong ball keeps rotating and is stable. Guess: the imbalance of the air outlets leads to the larger air volume on one side, which gives a rotating force to the ping pong ball.
Small size ball	
diameter	3cm
Maximum height from air outlet (center of circle)	14.5cm
Minimum height from air outlet (center of circle)	12cm
Average height from air outlet (center of circle)	13cm
Horizontal deviation	1.5cm
Medium size ball	
diameter	5cm
Maximum height from air outlet (center of circle)	7.5cm
Minimum height from air outlet (center of circle)	6.5cm
Average height from air outlet (center of circle)	7cm
Horizontal deviation	<1cm
Large ball	
diameter	6.5cm
Maximum height from air outlet (center of circle)	6cm
Minimum height from air outlet (center of circle)	4cm
Average height from air outlet (center of circle)	5cm
Horizontal deviation	<1cm

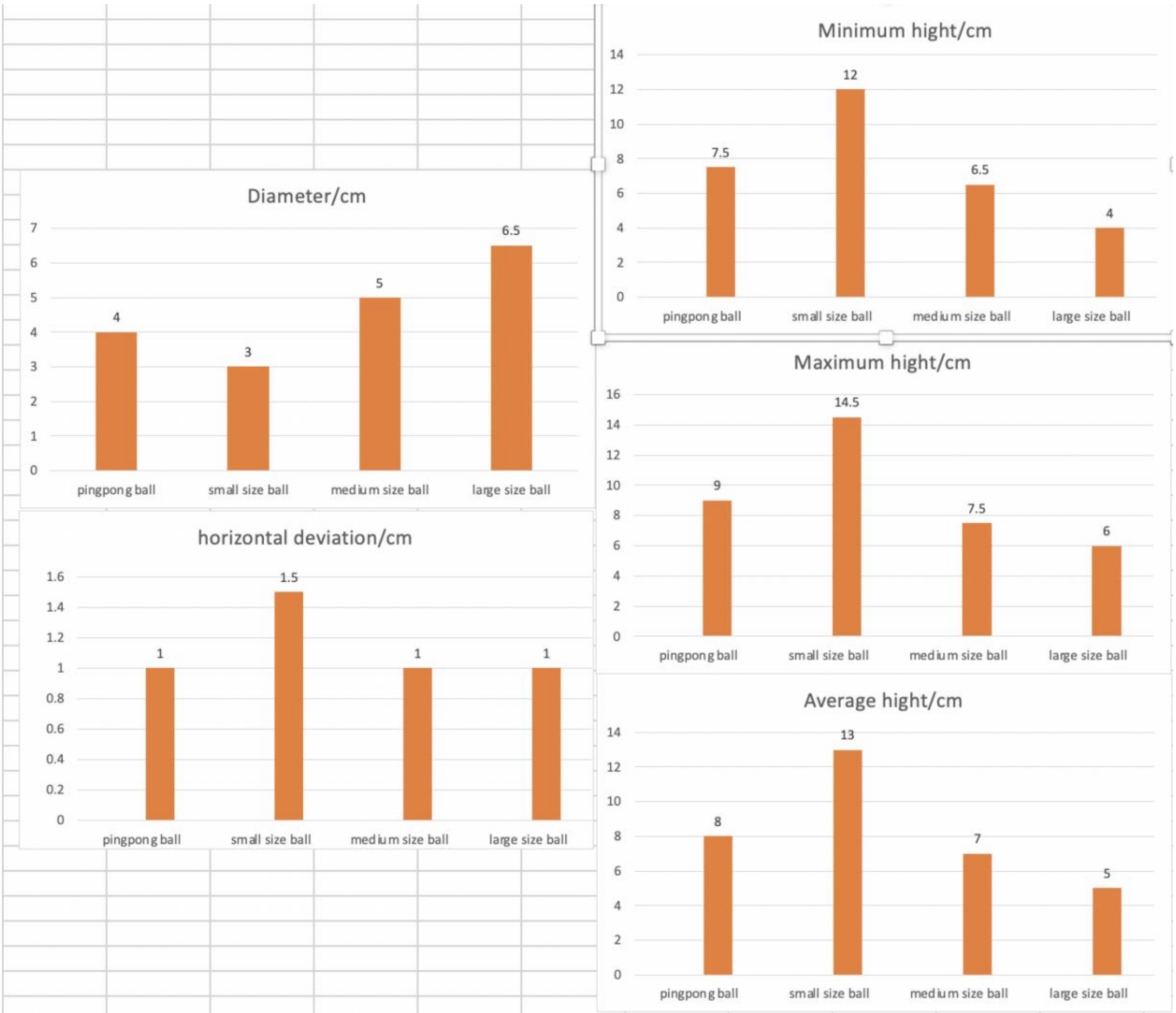


Figure 7. Quantitative comparative analysis of experimental data.

Another quantitative comparison of this group of experiments is shown in Figure 7. From this picture we conclude that in the Chinese experiment the smaller the ball the higher it can fly because of the lower gravity, but also because of the lower weight it is more susceptible to disturbance and therefore more unstable. The larger ball, on the other hand, will fly a smaller distance but will show less stability, i.e., less horizontal deviation, and a smaller difference between the maximum hight and the minimum hight.

There are some problems that should be avoided in further experiment:

In typical experiment: the experimenters did not try multiple wind modes, and did not compare balls of different sizes. In additional experiments, the modelling of the blower made it impossible to use the formula to calculate, and there was also no comparison of different wind forces (both experiments were because the weak wind mode could not support the ball). However, combining the two experiments, it is still able to get a complete analysis.

2.3.2. Glide experiment

In this experiment, a glider plane flying in equilibrium slowly sinks to the ground. The glider descends at an angle θ, which is determined by the balance between the glider mass, gravity, and the lift and drag forces experienced by the glider. The goal of the lab is to measure the lift and drag ratio of a glider and see how it varies as the wing geometry changes.

Consider the free-body diagram in Figure 7. The drag force acts in the direction of motion, while the lift force acts in the direction perpendicular to the glider motion. The weight of the glider acts vertically.

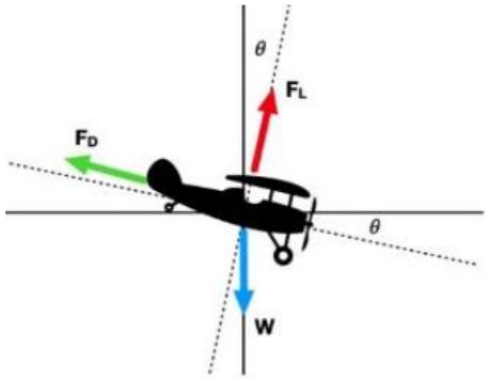


Figure 7. Free body diagram of a glider [5].

Projecting FD and FL onto the horizontal and vertical directions, there are:

Vertical:

$$F_L \cos\theta + F_D \sin\theta - W = 0 \tag{10}$$

Horizontal:

$$F_L \sin\theta - F_D \cos\theta = 0 \tag{11}$$

As a result,

$$\frac{F_L}{F_D} = \frac{C_L}{C_L} = \frac{1}{\tan\theta} \tag{12}$$

Therefore, CL/CD can be inferred by measuring the angle θ. In the laboratory, θ can be measured for paper-planes experimentally. Paper-planes have triangular-shape wings, so it is suitable to measure wing length and open angle, deducing its geometry.

In the laboratory, gliding paper-planes with different wing geometry are launched with the same velocity in order to make sure the initial conditions are the same. To reduce experimental deviation, each step is repeated 5 times, and then software Tracker can be used to track the descend angle.

Procedure:

1. Choose a test area that does not have any strong winds and fold several identical paper-planes.
2. Weigh the planes. (In the experiment make planes in A4 paper, whose weight can be found on the Internet.)
3. Launch a model so that it leaves hand level with the ground, at a consistent speed. Try to launch with approximately zero incidence angle and adjust the position of the wings and tail so that the glider flies smoothly.
4. When launching the model, record the initial flight with a camera. Make sure to have a clear contrast between the plane and the background, and the camera should be positioned stably and horizontally.
5. Repeat this step 5 times. Try to launch the model with the same speed, at the same height and with the same launch angle each time.
6. Change the wing planform’s geometry with different folding methods.
7. Repeat the previous experiments taking another 5 test flights and measuring the glide angle.

Experimental results:

The graphs below in Figure 8 are the results of one of our tests. Other graphs are somewhat similar as these, so these two graphs can show the analysis generally. The first graph shows the position of plane model in a time frame. The second graph shows the ratio of CL to CD. The angle began recording as soon as the plane started descending at a stable direction (for example, counting from the 10th point).

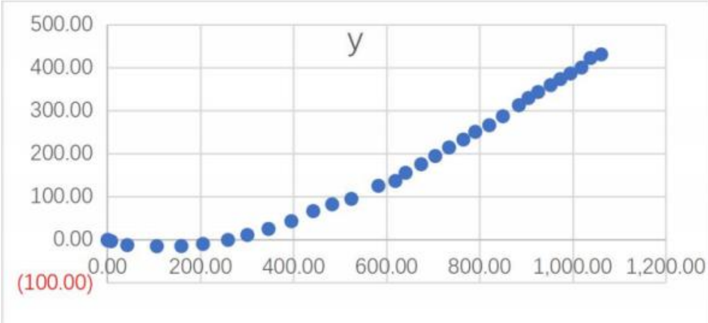
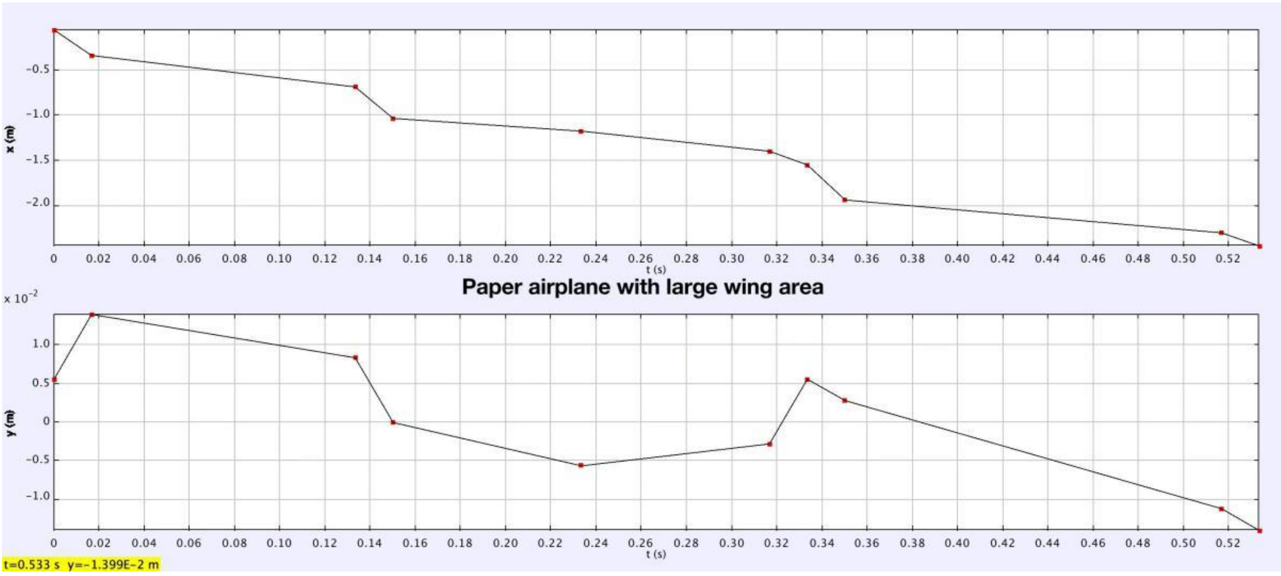


Figure 8. Graph of position [6].

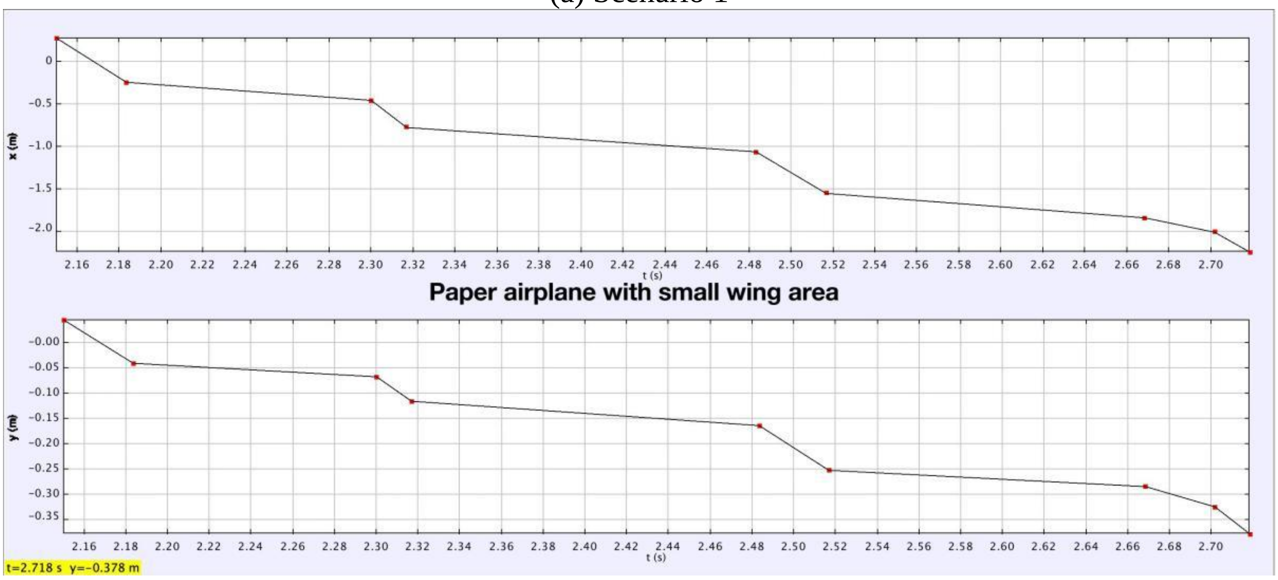
Wide wing's CL/CD is 4.5, and narrow wing's CL/CD is 2.2. In conclusion, the lift to drag ratio has direct relationship to the wing size. As the wing becomes larger, CL/CD increases. The reason may be that with larger wing, the larger area to meet the air, so the life force is larger.

Further discussion:

The situations are shown in the graphs below in Figure 9. The first graph shows the position of a paper-plane with a larger wing area, and the second one shows the position of a paper-plane with a smaller wing area.



(a) Scenario 1



(b) Scenario 2

Figure 9. Position of a smaller wing-area paper plane.

There are several situations where the plane doesn't fall smoothly, and possible causes are as below:
1. Either turn left or right: possibly because the models are made asymmetric.
2. Fall down too quickly: the center of mass is at the front of the plane.
3. Suddenly rise then descend: it is "stall" The reason of the rise may be either the wings are not plain or the initial speed is too large, making the lift force too large.

3. Application of aerodynamics in civil cars and racing cars

Aerodynamics is a branch of fluid mechanics. The faster the car runs, the greater the resistance. In the resistance calculation formula $F_{\mathrm{d}} = \frac{1}{2}\rho \mathrm{u}^2 c_{\mathrm{d}} A$, $\mathrm{F_d}$ is the drag force, which is by definition the force component in the direction of the flow velocity. ρ is the mass density of the fluid. u is the flow speed of the object relative to the fluid. c_{d} is the drag coefficient. A is the reference area. the increase of speed u will lead to the increase of resistance in the exponential square, while the pressure drag coefficient c_{d} is a linear influence, which also means that the role of aerodynamics will be greater and more important in modern civilian cars and racing cars with faster speed.

3.1. Comparison between the early stage automobile design and modern automobile design

We can see that in the early automobile design, that is, in 1920, the automobile pressure drag coefficient reached 0.95. However, it is only 0.30 among modern civil vehicles. In 1920, the shape of the car was plate, but now it is oval, which is the streamline of ordinary people. Under the calculation of the resistance formula, this means that the air resistance is reduced by 65% under the conditions of the same speed, the same force area and the same air density.

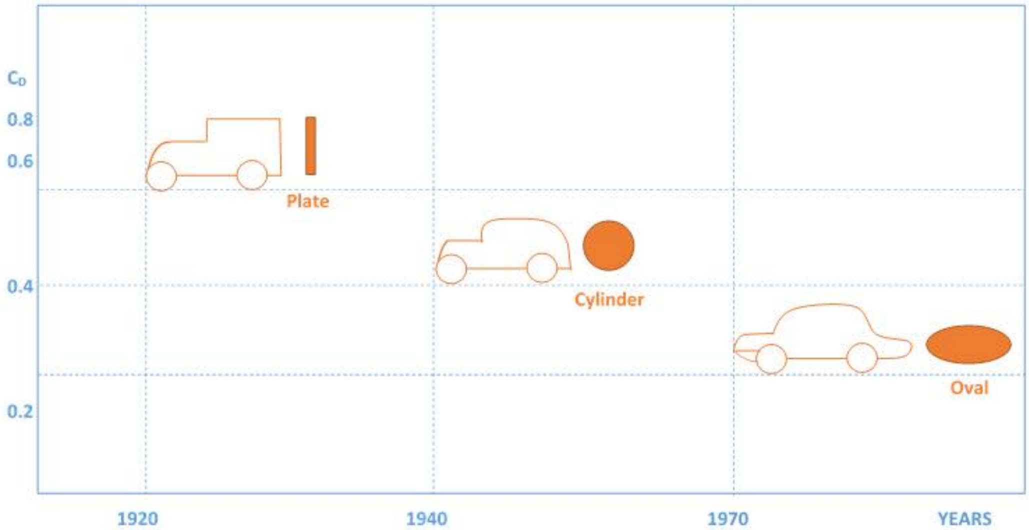


Figure 10. Time history of cars' aerodynamic drag in comparison to change in geometry of streamlined bodies (blunt to streamline) [4].

There is no doubt that such achievements are obviously huge. Under the influence of less resistance, we can obviously save more energy against resistance, which is also contributing to slowing down global warming. Compared with blindly not using fossil fuels, the application of this technology can save more energy consumption effectively and reasonably. At the same time, today's cars and racing cars have a greater impact on the smaller Cd. As mentioned above, the impact of speed on resistance increases in square. Today's cars are faster and racing cars can reach a speed of more than 300km/h. Therefore, in the drag force calculation formula for air resistance, we can understand how much speed affects resistance, So this is why the air resistance and its shape were not paid attention to when the automobile was just invented, and the air resistance became more and more important as the automobile became faster and faster. It is precisely because now almost all automobile manufacturers have their own aerodynamic departments and wind tunnel devices to simulate the direction and size of the resistance when the automobile is driving.

The direction mentioned here also means that the resistance is not only the size of the most important, but also how to apply this resistance to a more appropriate place is a very important technology and research direction. Especially in the aerodynamic application of the car, the direction of resistance has a more obvious effect on the performance of the car, and some aerodynamic packages also prove this to everyone more intuitively.

3.2. The role of the rear wing of racing cars and DRS devices in F1 racing

There are often some huge rear wings in the racing car. These rear wings do not make the car look more beautiful, but have a very significant impact on the performance of the car. If the car needs to make a faster lap speed on the track, it means that it needs to have a faster speed on the track to pass through the curve, and when the car passes the curve too fast, it will generate too much centripetal force, which will cause the car to have understeer, and there is no way to pass the curve faster, which requires the car to generate enough downward pressure to catch the track. There are many methods here. First, the mechanical downforce of the car itself is the weight of the car itself. Second, the grip of the tires is the viscosity generated by the tires and the ground. Finally, aerodynamics generates downforce with the Spoilers in various parts of the car at high speed through different flow velocities.

Figure 11. The huge rear wing in the racing car [10].

The rear wing of a racing car can be seen as an inverted airplane wing. The airplane wing can provide lift for the airplane to fly in high altitude. The rear wing in the same racing car can also provide a downward pressure for it. But there is also a problem. When the wing generates lift or the real wing generates downward pressure, it will also generate resistance. Resistance is the natural reaction of air resistance through racing. Resistance will slow the car down. Therefore, more downforce is good, but too much downforce will also produce too much resistance, which will have a negative impact on the performance of the car. Very high performance sports cars, such as Le Mans or F1, have a ratio called lift drag ratio. Car designers try to maximize this, so that cars have enough force to go around corners, but not too slow. Indy cars and cars of similar design can achieve 3G, that is, three times of its own weight downforce at the speed of 200 mph. This means that they can hang upside down on the track. As long as they keep fast enough, they will have enough downforce to pass the curve at high speed.

The car needs more downforce to maintain the stability of the tail in the corners, which is a positive effect. The downforce in the straight line is instead the resistance of the car, which will make the car slower in the straight line and have a negative impact. Because of this, there is a very smart design in the F1 car, namely the DRS system, which can change the angle of the real wing of the F1 car, so that the downforce generated by the F1 car in the straight line will be smaller and the downforce will be higher in the curve, This seemingly simple system can help the car's top speed in the track increase by 5-7 km/h. Of course, the significance of this system is to overtake and increase the enjoyment of the game. It is not available throughout the game but has certain triggering conditions. These adjustable real wings, namely active aerodynamics, are also used in some road sports cars.

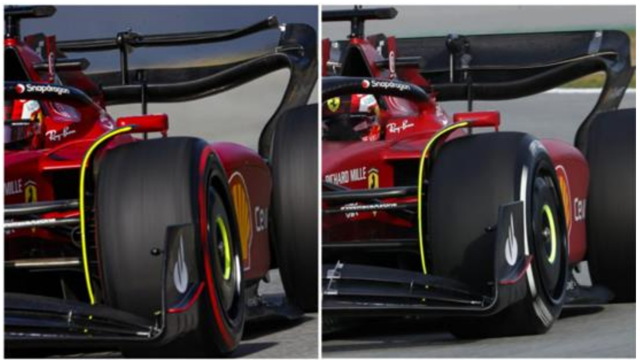


Figure12. DRS system on and off [8].

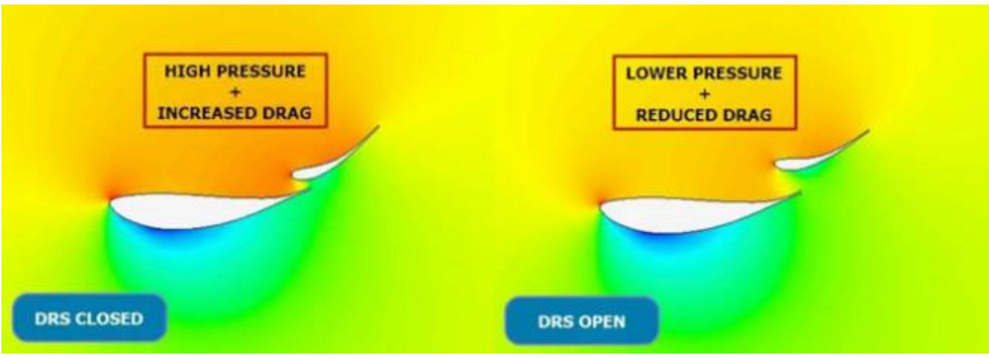


Figure 13. Principle demonstration of DRS system [8].

3.3. Ground Effect in F1 Racing and "Porpoising Problem"

In F1 racing, not only real wing can generate downforce, but also a lot of aerodynamic kits can generate downforce. These parts are called spoilers. There are many small spoilers and huge real wings on the racing car, which also form the aerodynamic balance of the whole racing car, There will be a lot of fine tuning and setting available for different racetracks and driver teams with different driving styles. In addition to the downforce caused by real wing, the design of ground effect is open to teams in the 2022 F1 race to reduce the difficulty of car following.

First of all, ground effect refers to the application of Bernoulli principle to make the car's aerodynamics become a large spoiler. In addition, they also added some skirts on the side of the car to seal the low-pressure area. It's not like F1 cars with ground effect disabled have many parts. Lotus 78 was the first car to introduce ground effect. This car was designed in 1976, which completely changed F1 forever. The team began to introduce skirts at the side of the car to close the floor and prevent high-pressure air from leaking from underneath the car. These skirts are placed as low as possible for maximum efficiency, so they skim the ground and have a small suspension to suit the track. However, because the track has many irregularities, potholes and curbs, one or two skirt edges may rise, which will cause pressure changes under the car, thus reducing downforce. This may cause the car to lose balance or even turn over, because there are two different pressures, which are very dangerous. And a lot of riders lost their lives, so the FIA realized that the ground effect technology was not mature enough and extremely dangerous, so it was disabled in 1983.

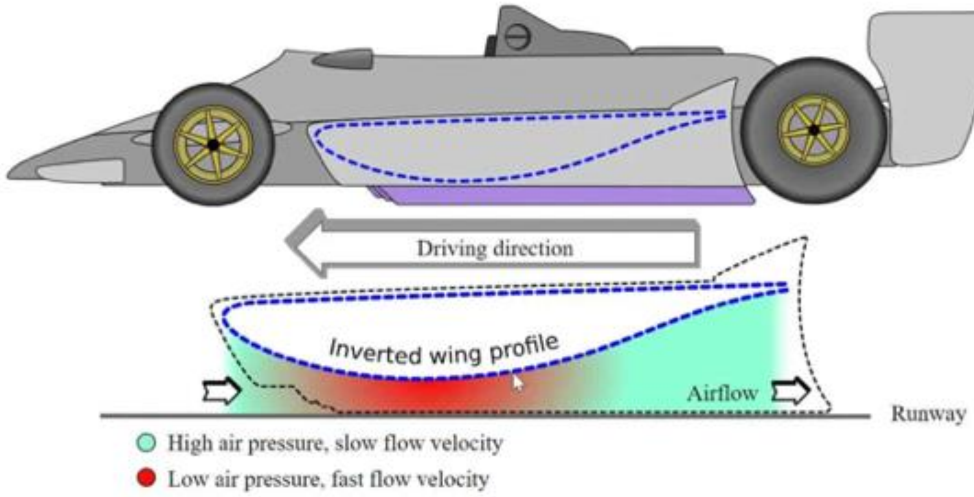


Figure 14. F1 Racing Ground Effect Principle [4].

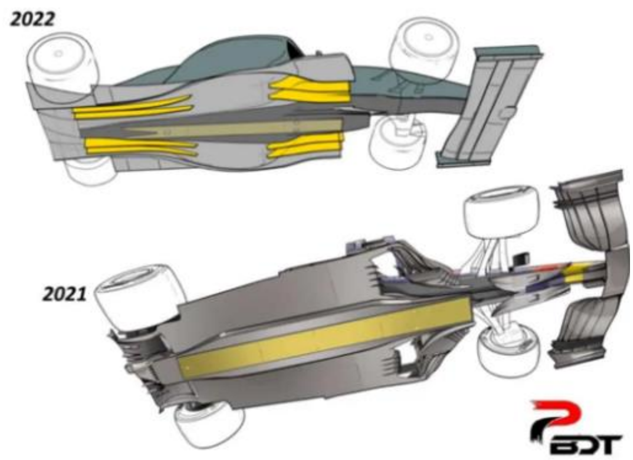


Figure 15. Design comparison of F1 car before and after using ground effect [8].

However, the downforce generated by the use of the spoiler in the F1 team today will generate turbulence. These small fins on the F1 car will make the air unstable, which means that it will cause difficulties in car following, because the car behind will not be able to play all the aerodynamic effects of the car after encountering these turbulence, and will lose a lot of downforce, It is estimated that the car behind may lose up to 50% of the downforce when following the car in front, and there will be less turbulence after using the ground effect because more downforce is generated by the floor, that is, the ground effect. It is estimated that only 18% of the downforce will be reduced. This can greatly increase the enjoyment and competition of F1. At the same time, today's technology is mature enough and has strict design control, which means that the tragedy caused by ground effects will not happen again. However, the ground effect technology still has a difficult problem for each team to overcome, which is the "Porpoising Problem".

The new generation of ground effect aerodynamics vehicle of equation one has brought a new phenomenon. Porpoising, otherwise known as bouncing, is the motion today's generation of F1 cars make when their under-floor aero stalls and the car is pushed away from the ground. Like a dolphin or porpoise moving through the water, up and down along the length of their body, so too do the F1 cars. But unlike with the dolphins, it is not pleasant to watch and even worse to experience. The principle is that when the current car has ground effect, the downforce generated by excessive speed is too large, resulting in that the car floor is almost completely attached to the track and the air cannot pass through, so the car body bounces up at a moment's loss of downforce, and then the car floor and track have enough space for the air to pass through, but the last process is repeated quickly, This leads to the problem of porpoising in the racing car and repeated jumping at the straight end of the track.

4. Current technology and aerodynamic solutions

Currently, the modern F1 car underbody tunnel is closer to a “Venturi” type, unlike those early ‘wing style’ F1 side pods. The car at the top in Figure 15 is 2022 F1’s simplified concept car, a tunnel shape with a throat, nozzle, and diffuser. Regarding to unstability of porpoising, for example, tunnels are far safer than diffusers as flat floors against diffusers suffer from much higher pitch sensitivity than tunnels. In spite of this, any underbody device can be prone to porpoising and other aerodynamic issues. With the growing accessibility of basic Computational Fluid Dynamics (CFD) aerodynamics simulation testing, F1 car designers are more able to utilize both in-house super-compute CFD and physical wind tunnel test facilities.

At any rate, there exists some difficulty in replicating low ride heights during aerodynamic development. To solve this problem to the greatest extent possible, modern wind tunnels utilize a moving belt floor and a model suspended from the roof by a device called a sting. The rotating tires are supported from the sides or by attachments to the car that isolate it from the belt vibrations. The model of the car is equipped with sensitive instrumentation inside the model. Real-world racing is, by contrast, very different in that the car is not held between static positions, and tunnel testing faces challenges in recreating dynamic vehicle conditions. Nonetheless, tunnel testing is still a mature technology and will continue to evolve its accuracy.

CFD uses well-established formulas to predict the behavior of air around an object. It breaks down the air volume around a vehicle into a series of units of various sized discretized volumes known as the “mesh”. The mesh density is carefully controlled, not just for accuracy vs compute compromises but also to ensure a stable result across all tests. Again, CFD simulations are typically static assessments of a vehicle performance. But with the rapid development of computer technology, computers can process more complex information in larger quantities, making CFD simulation closer to reality in the future.

5. Conclusion

The fun and the ratings of racing lie in the competition generated by the tangle of multiple cars and the near-identical speeds. Just as the DRS system was devised, the DRS system is designed so that a vehicle can increase the likelihood of overtaking in a range where overtaking is possible. The system is only used when the car behind is only one second away from the car in front and can only be used in a specific area. This is the same as the ground effect being allowed to return in the FIA this year, as previously F1 cars had a large number of aero kits on the move to provide downforce on the car itself, and these aero kits were only providing an advantage to that one car itself, and the complete opposite for the rear car. As the ratio of aerodynamic advantage compared to other components is getting higher, all teams are looking for better aerodynamic packages, which means that aerodynamics is especially important for an F1 car. But these aerodynamic packages create dirty air, or turbulence, after passing clean air. Dirty air is unstable air that does not provide enough downforce for an F1 car, which means that following is more difficult in F1 and overtaking is more difficult, the closer you get to the car in front of you, the greater the effect of dirty air. This is why the FIA has reintroduced the ground effect this year, which aims to make cars more dependent on mechanical downforce, or gravity, to generate downforce, and the FIA wants teams to use fewer aero packages, or spoilers, on their cars to generate downforce, as these produce a lot of dirty air.

Instead, more of what is known as the Venturi effect is created by allowing air to flow underneath the car by way of two channels of reduced diameter. As the airflow contracts, the airspeed increases and the air pressure decreases. Although this results in a final downforce that is less than that generated by using the Spoiler it provides more overtaking possibilities making the whole race more exciting. Finally, with regard to the Porpoising Problem, the FIA has decided to analyse the size of the bounce in order to establish a uniform standard to regulate the safety of F1 cars, as the bounce is too violent and may even affect the driver's spinal health and brain health. The main cause of the Porpoising Problem is the proximity of the chassis to the track, so raising the chassis is the easiest solution, despite the loss of speed. It was only the first year of the return of the ground effect and the first time the cars were driven on a real track that the Porpoising Problem was discovered too late, but over time the F1 engineers have been able to collect more data and study it. The Porpoising Problem should be solved for every team without any loss of speed, following the cars will become easier, and F1 racing will become more and more intense to watch and race.

References

[1] Wang M., Liu Z X., Huang S Q. (2005) The application of FLUENT software in the car styling. Machinery Design & Manufacture, 20: 105 - 109.

[2] Hang X., Wang H., Hou W. (2016) Design of an Aerodynamic Package for FSAE Racing Car and CFD Analysis. Experiment Science and Technology, 20: 116 - 119.

[3] Amend J M. (2008) Volt's Exterior Design Extends Range - GM says aerodynamics play a greater role than reducing mass. Ward's auto world, 8 (1): 44.

[4] Fuller. (2004) The aerodynamics of Formula One car cooling ducts. University of Cambridge, 22: 104 - 109.

[5] Chaudhry S R. (1994) Numerical experiments on the aerodynamics of waveriders. University of Glasgow (United Kingdom).

[6] Sun J., Sun Y. (2022) Wind tunnel test method and drag reduction research of automotive aerodynamics. Automotive Abstracts, 22 (10): 41 - 44.

[7] Wu Q., Cao Y., Li F., Zhu X., Jiang X., Lv Q. (2022) Design and flow field analysis of FSC racing aerodynamic package. Automotive Practical Technology, 47 (18): 98 - 102.

[8] Ding P., Yang H., Gao W., Deng Z. (2022) Numerical Simulation of the Aerodynamic Characteristics of the Rear Wing of a Car and Analysis of Its Influencing Factors. Tractors and Agricultural Transporters, 49 (4): 19 - 24.

[9] Zeng F. (2022) Application and Exploration of Virtual Simulation Experiment in the Teaching of "Automobile Aerodynamics". Science and Technology and Innovation, 34 (14): 31 - 33+37.

[10] Qin Q., Liu X., Wang H., Wang X. (2022) Research Progress on Aerodynamic Performance of Vehicle Queue Driving. Automotive Practical Technology, 47 (8): 188 - 204.

[11] Chen Y., Hu S. (2022) Aerodynamic package design of university student formula racing. Journal of Changshu Institute of Technology, 36 (2): 82 - 86+115.