Analysis of Aerodynamic Structure of Formula One Wake

Zhu Shizhuo

Wuhan University of Technology, Wuhan, 430070, China

Keywords: formula racing; tail; diffuser; Ansys simulation; flow field analysis

Abstract: This paper introduces the aerodynamic characteristics of the Formula 1 car and the effects of the rear negative lift, emphasizing the importance of its wake utilization, thus extending the two aerodynamic packages of the tail fin and diffuser on the Formula 1 car. It details the functions of the two to enhance the lift, the principle of using the wake and its development in Formula One, and demonstrates the importance of the tail and diffuser in automotive aerodynamic applications. The aerodynamic design of the car is instructive.

1.Introduction

Aerodynamics is the key to success of formula one race. [2] Pneumatic designers have two primary concerns: first, the manufacturing of the lower pressure makes the race car closer to the surface while improving the turning ability; second, the air resistance caused by the air vortex to slow down the vehicle speed is minimized. In fact, the drag coefficient of a typical F1 car is around 0.7 Cd, which is much higher than an ordinary car, because most of the resistance is used to convert to downforce and reduce aerodynamics lift. Since the grip of the F1 is more than grip, the rear lift has a great influence on the dynamics and steering stability of the Formula one car.

The diffuser is a large shovel-shaped component located at the rear end of the car floor. The diffuser helps to make the bottom of the car similar to the shape of the lower part of the tail. [2]The diffuser creates a large void in the rear of the car, which pumps the bottom airflow and accelerates the bottom airflow, which greatly reduces the pressure on the bottom of the car and increases the downforce of the car. This effect is called "ground effect."

2. Introduction to the aerodynamic characteristics of the F1

2.1 Negative lift generation principle

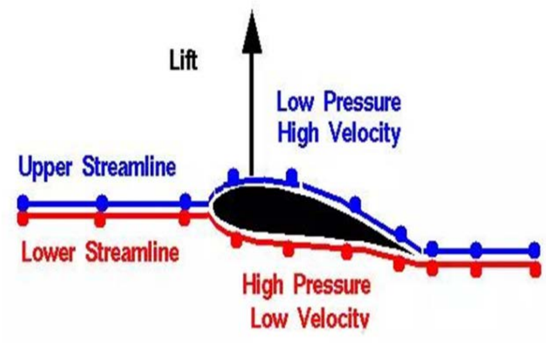

Fig 1 Bernoulli principle

$$P_1 + \rho g y_1 + \frac{1}{2}\rho v_1^2 = P_2 + \rho g y_2 + \frac{1}{2}\rho v_2^2 \quad (1)$$

By Bernoulli's equation (conservation of fluid mechanical energy): kinetic energy + potential energy + pressure = constant of the fluid. [1]The pressure potential energy of the fluid, the gravitational potential energy, and the sum of the kinetic energy are a constant, which is an expression of energy conservation. When the air flows through the asymmetric airfoil on the upper and lower surfaces, the surface velocity of the long path is high, so the pressure difference is formed on the upper and lower surfaces under pressure.

2.2 How the tail wing works

[3]The empennage creates a downforce through the pressure differential between the top and bottom of the body. This difference in air pressure is due to the way the air flows through the empennage. According to the Bernoulli principle, for a gas of a certain flow rate, the higher the flow rate, the lower the gas pressure. The empennage creates a pressure differential by moving air molecules from the leading edge to the trailing edge in different ways. The longer portion of the empennage requires a high velocity (low pressure) of airflow on that side to achieve a low velocity (high pressure) airflow at the top of the end of the empennage.

The low pressure region at the bottom of the empennage causes the top high pressure region to exert a downward thrust on the empennage, which can create downforce and reduce aerodynamic lift. [4]The main parameter affecting the aerodynamic characteristics of the tail is the tail angle of attack. When the angle of attack of the tail increases within a certain range, the downforce will also increase. However, when the angle of attack increases to a certain value, airflow separation occurs. When the airflow separation occurs, the upward momentum of the airflow is reduced as compared with the state in which the separation does not occur, so that the generated downforce is drastically reduced, and the generated resistance is also significantly increased. This is called stalling.

3. Simulation of the tail of a modern F1

3.1 Model establishment

We use the latest modern Formula 1 racing car to analyze the external flow field of the tail section

3.2 Calculation domain setting

In this simulation process, the calculation domain is a cuboid with 10 times the length of the car (6 times before 3 times), 4 times the height of the car and 9 times the width of the car.

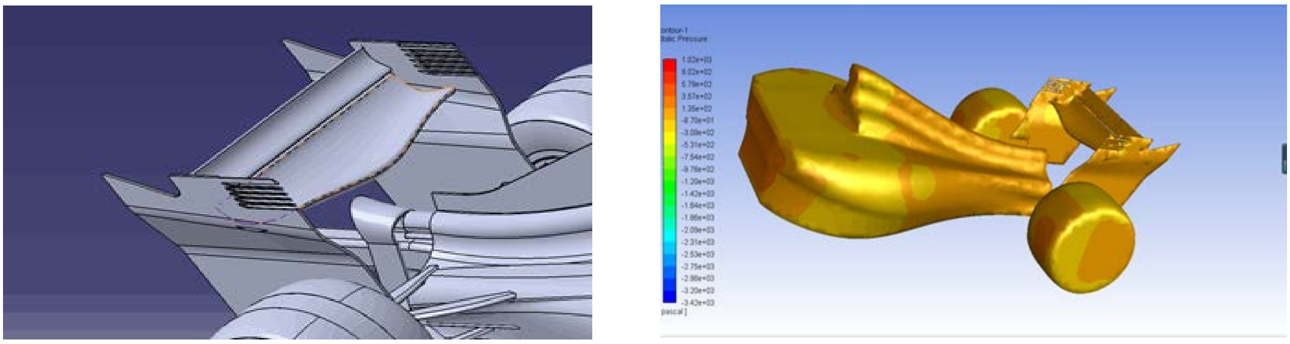

Fig 2 tail wing model

3.3 Grid division

In the meshing module of Ansys software, we have meshed the two models. The entire computational domain uses an unstructured tetrahedral mesh, and the mesh is encrypted at the front end, the A-column, and the rear view mirror, and the mesh gradually transitions from small to large to the wind tunnel wall. The total number of grids is 875,946, and the total number of grids in 911 is 536,901.

3.3.1 Boundary condition setting

This article sets the ambient pressure to a standard atmospheric pressure. Establish a digital wind tunnel based on real wind tunnel measurements. The wind direction is positive along the x-axis, the end face at 3 times the length of the front of the car is the entrance boundary, and the end face at 6 times the length of the tail is the exit boundary.

Table 1 boundary conditions

Boundary parameter	Set value
Import speed / (m / s) - 1)	41.6
Outlet pressure	Standard atmospheric pressure
Speed / (m / s)	41.6
Ground boundary	Sliding ground
Body wall	No-slip
Turbulence intensity	0.5
Dissipation rate	0.5

3.3.2 Solving models and parameter settings

[2]K-epsilon is one of the turbulence model theory, referred to as the k-ε model. The k-ε model is the most common turbulence model based on the turbulent energy k and the turbulent dissipation rate epsilon. This is a very popular two-equation model, reliable, good convergence, low memory requirements, and many variant models, such as Standard, RNG, Realizable, etc.

Since the flow field of the vehicle flow problem is generally a constant, isothermal, incompressible three-dimensional flow field, its complex shape will cause the separation of the airflow and should be treated as a turbulent flow. This paper uses the standard k-epsilon model to simulate. In the simulation, we set the fluid to air, the solid part is the aluminum part, the viscosity model is set as described above, and the solver is set to simple mode.

3.4 Analysis of the external flow field of F1

Through simulation, the following results are obtained.

3.4.1 Racing tail flow spectrum

[6]The flow spectrum refers to the family of streamlines in which the airflow flows. The tangential direction of any point on the streamline is the velocity direction of the airflow micelle at that point. When studying the internal and external flow spectrum of a car, it is generally assumed that the air is a constant flow, that is, the flow parameters do not change with time

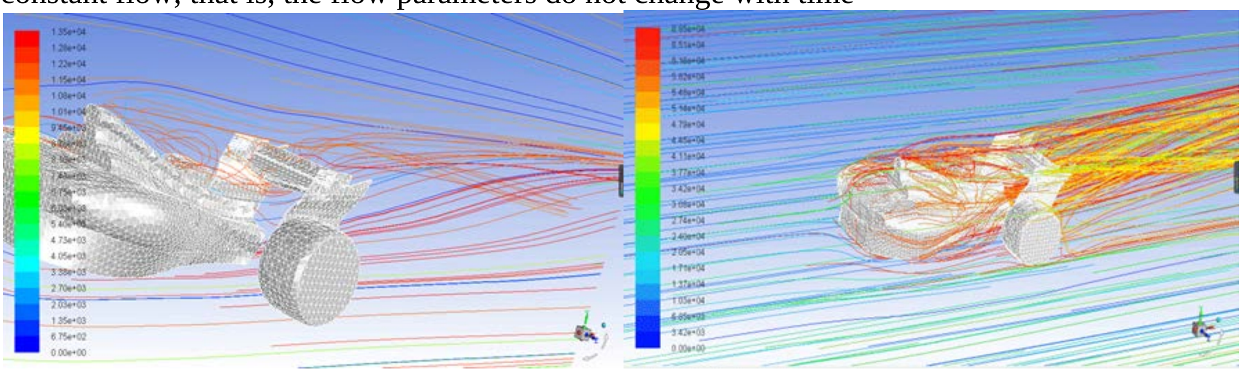

Fig 3 tail flow spectrum

It can be seen from the tail flow spectrum of the car that the flow line of the modern F1 car is clear and the vortex area is smaller, so the low pressure zone is smaller, the air resistance coefficient is smaller, and the negative lift is larger.

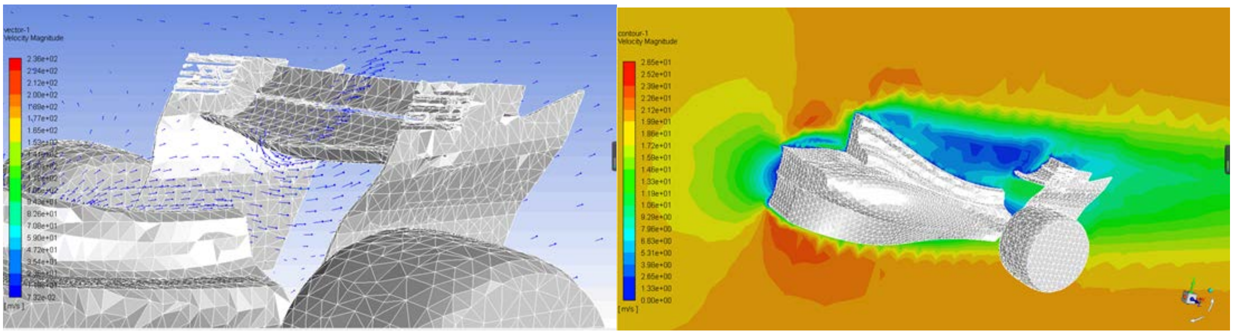

Fig 4 speed vector and speed cloud map

3.4.2 Racing tail speed vector illustration

As can be seen from the figure, the front negative lift wing divides the airflow into upper and lower parts. The upper part of the airflow flows through the upper surface of the vehicle body, the lower part flows along the chassis, and the speed is significantly accelerated. After the upper surface airflow passes through the negative lift wing, it will A positive pressure zone is formed on the airfoil surface, resulting in greater aerodynamic drag, but with multiple airfoils, it produces a greater negative lift, and the upper and lower airflows rejoin in the wake of the car. From the tail airflow distribution, it can be seen that the counterclockwise vortex at the top of the rear negative lift wing and the clockwise vortex near the ground, the rotation of the vortex and the friction of the airflow require a lot of energy, thus forming a strong negative pressure zone at the rear of the vehicle. Pneumatic lift also produces differential pressure resistance.

3.4.3 Pressure cloud at the rear of the car

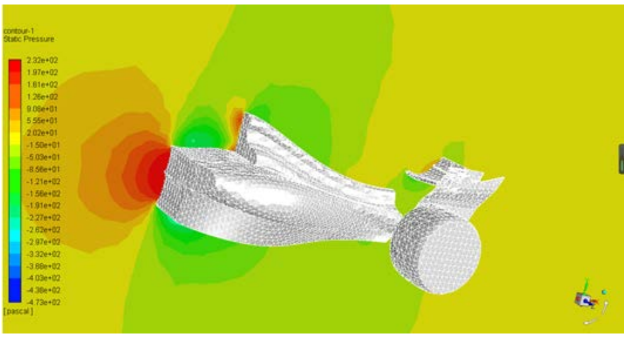

Fig 5 pressure cloud map

It can be seen from the figure that the upper surface of the negative lift wing after the racing is a positive pressure zone, and the negative pressure zone is formed in the chassis due to the separation of the airflow, which will increase the aerodynamic lift of the vehicle and the gas flow rate at the tail is faster. The negative pressure is also stronger, increasing the aerodynamic lift. However, at the junction of the negative pressure zone and the positive pressure zone, the airflow separation will form different degrees of vortex, causing the airflow to rotate and fall off, thereby consuming a large amount of energy and increasing the aerodynamic drag.

4. Racing diffuser

4.1 The influence of the bottom flow field on the aerodynamic characteristics of the car

[5][6]When the car is driving, it will generate three airflows, one from the airflow passing over the car body and the rear wing, and the airflow from the side through the radiator, more importantly, the airflow through the chassis, because this road. The airflow is not only an important condition for forming the downforce, but it also accounts for more than 45% of the total airflow above and below the vehicle body. For this reason, the bottom design of the f1 car is strategically of great importance. In order to improve the aerodynamic performance of the bottom of the car body, the diffuser installed

under the rear of the car becomes an important aerodynamic package. The basic principle of generating downforce with the tail is the same. The air passes under the airfoil longer than the distance above, causing the air below to accelerate, the air pressure to drop, and the pressure difference between the upper and lower surfaces. This is called the downforce.

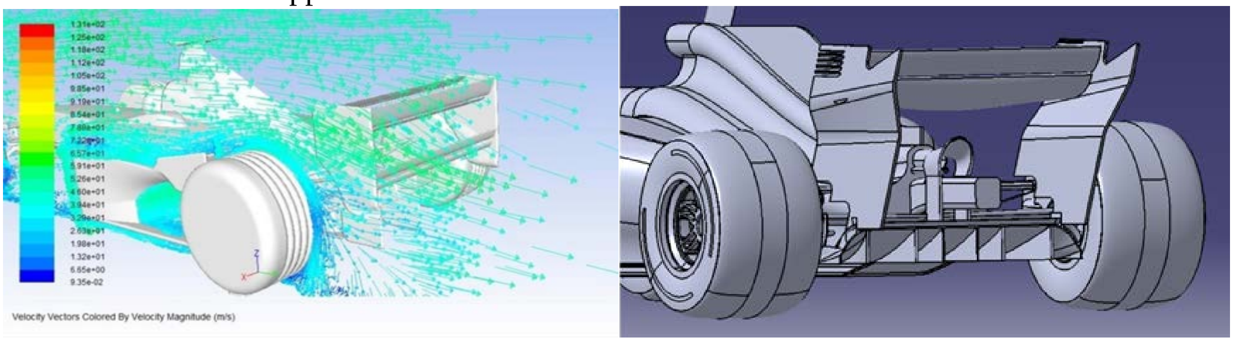

Fig 6 diffuser model and speed vector

4.2 External flow field analysis of diffuser

As can be seen from the figure, the flow rate of the air flowing through the bottom is obviously accelerated, the pressure at the bottom of the car is higher than that at the bottom, and the bottom airflow enters the wake to form a strong backflow, forming a lower vortex, which forms a strong negative at the rear of the car. Pressure zone. The function of the diffuser on the car is to speed up the airflow at the bottom of the car, reduce the pressure, create a larger pressure difference between the upper and lower surfaces, and bring more downforce and pneumatic grip.

5. Conclusion

From the theoretical and simulation aspects, the function of the tail fin and the diffuser is verified. The tail and diffuser can significantly increase the downforce of the formula car under the good optimization design, and improve the vehicle's power and handling stability. The development of the empennage and diffuser is a process of continuously applying the aerodynamic performance of the wake of the car to the extreme. Although there are rules and restrictions, the ultimate direction of the effort is to make good use of the wake of the car. Take performance as much as possible.

References

[1] Gu Zhengqi. Automotive Aerodynamics [m]. Beijing: China Communications Press, 2005, 116-191

[2] Fu Limin. Automotive Aerodynamics [m]. Beijing: Mechanical Industry Press, 2006, 16-27

[3] Gu Zhengqi. Modern technology of automobile body [m]. Beijing: Mechanical Industry Press, 2009, 27-32

[4] Huang Xiangdong. Automotive Aerodynamics and Body Modeling [m]. Beijing: China Communications Press, 2004, 168-170

[5] Jiang Tao. Research on optimization of aerodynamic design of automobile body [d]. 2011.

[6] Ma Yong, Zheng Weitao, Han Jiurui, et al. Application of Computational Fluid Dynamics in F1 Motorsports[J]. Journal of Wuhan Institute of Physical Education, 2005, 39(3): 52-55