DESIGN AND SIMULATION OF AERODYNAMIC WINGS OF FORMULA ONE RACING CAR

Pritam Pain[1], Deep Dewan[2], Arighna De[3]

[1]Dept. of Mechanical Engineering, University of Engineering & Management, Kolkata, West Bengal, India
[2]Dept. of Mechanical Engineering, Kingston Polytechnic College, Barasat, West Bengal, India
[3]Dept. of Mechanical Engineering, University of Engineering & Management, Kolkata, West Bengal, India

---

Abstract: The aim of this report is to introduce the design and simulation of aerodynamic wings that is generally used in formula one racing cars. In this paper, the basic concept of aerodynamics, related terms are described. From the design of aerodynamic wings to the simulation of the wings using Solidworks 2016 well described in this paper.

Key Words: Aerodynamics, Wings, Solidworks, Simulation

INTRODUCTION:

Aerodynamics is the study of motion of air, particularly as interaction with a solid object, such as an airplane and automobile wing. It is a sub-field of fluid dynamics and gas dynamics, and many aspects of aerodynamics theory are common to these fields. The term aerodynamics is often used synonymously with gas dynamics, the difference being that "gas dynamics" applies to the study of the motion of all gases, and is not limited to air. The formal study of aerodynamics began in the modern sense in the eighteenth century, although observations of fundamental concepts such as aerodynamic drag were recorded much earlier.

AUTOMOTIVE AERODYNAMICS:

Automotive aerodynamics is the study of the aerodynamics of road vehicles. Its main goals are reducing drag and wind noise, minimizing noise emission, and preventing undesired lift forces and other causes of aerodynamic instability at high speeds. Air is also considered a fluid in this case. For some classes of racing vehicles, it may also be important to produce downforce to improve traction and thus cornering abilities.

COMPARISON WITH AIRCRAFT AERODYNAMICS:

Automotive aerodynamics differs from aircraft aerodynamics in several ways.

First, the characteristic shape of a road vehicle is much less streamlined compared to an aircraft.

Second, the vehicle operates very close to the ground, rather than in free air.

Third, the operating speeds are lower(and aerodynamic drag varies as the square of speed).

Fourth, a ground vehicle has fewer degrees of freedom than an aircraft, and its motion is less affected by aerodynamic forces.

Fifth, passenger and commercial ground vehicles have very specific design constraints such as their intended purpose, high safety standards and certain regulations.

CONCEPT OF DRAG AND DOWNFORCE:

Motor sports are all about maximum performance, to be the fastest is the absolute. There is nothing else. To be faster power is required, but there is a limit to how much power can be put on the ground. To increase this limit, force to ground must be applied on the wheels. Increasing weight can do this, but weight makes handling worse and require more power. So, we need some virtual weight, we call it downforce and get it from airflow around the car.

Typically, the term "lift" is used when talking about any kind of aerodynamically induced force acting on a surface. This is then given an indicator, either "positive lift" (up) or "negative lift" (down) as to its direction. In aerodynamics of ground racing (cars, bikes, etc.) the term "lift" is generally avoided as its meaning is almost always implied as positive, i.e., lifting the vehicle off the track. The term "downforce", therefore, should always be implied as negative force, i.e., pushing the vehicle to the road.

Both the drag force and the downforce are proportional to the square of the velocity of a car. The drag force is given by:

$$F_{drag} = 0.5 C_d D A V^2$$

where is:
Fdrag - Aerodynamic drag
Cd- Coefficient of drag
D- Air density
A- Frontal area
V- Object velocity

Cd is the coefficient of drag determined by the exact shape of the car and its angle of attack. The downforce is given by:

$$F_{down} = 0.5 C_l A V^2$$

where is:

Fdown - Aerodynamic downforce
Cl- Coefficient of lift
D- Air density
A- Frontal area
V- Object velocity

Coefficient of lift, again determined by the exact shape of the car and its angle of attack. In the case of a modern Formula 1 car, the lift-to-drag ratio Cl/Cd has a typical value of, say, 2.5, so downforce dominates performance. In current motor racing competitions, including Formula 1, aerodynamic downforce plays the most important role in the performance of the cars. In the mid 1960's the use of soft rubber compounds and wider tires demonstrated that good road adhesion and hence cornering ability, was just as important as raw engine power in producing fast lap times. The tire width factor came as something of a surprise. In sliding friction between hard surfaces, the friction resistance force is independent of the contact area. It came as a similar surprise to find that the friction could be greater than the contact force between the two surfaces, apparently giving a coefficient greater than one. This would be similar to having glue on the tires, making them "stick" to the ground. The desire to further increase the tire adhesion led the major revolution in racing car design, the use of negative lift or 'downforce'. Since the tires lateral adhesion is roughly proportional to the downloading on it, or the friction between tire and road, adding aerodynamic downforce to the weight component improves the adhesion. Downforce also allows the tires to transmit a greater thrust force without wheel spin, increasing the maximum possible acceleration. Without aerodynamic downforce to increase grip, modern racing cars have so much power that they would be able to spin the wheels even at speeds of more than 160 km/h. Downforce, or negative lift, pushes the car onto the track. It is said that at maximum speed, an F1 car produces 5 g's of downforce! 5 times its weight pressing it down onto the track. Produced by almost every part of F1 car but mostly by use of diffuser and wings in the way that longer cord length is facing downward. Helping to induce more tire grip, but more downforce usually induce more drag. Downforce has to be balanced between front and rear, left and right. We can easily achieve the balance between left and right by simple symmetry, so it will not be discussed. Front and rear are a different thing. Flow in the front greatly affects flow in the back of the car, and vice versa. Downforce must be adjusted according to racing track and behavior of the car. Too much front downforce induce oversteer. Too much back downforce induce understeer. Variating downforce you can resolve the problems with oversteering or understeering car.

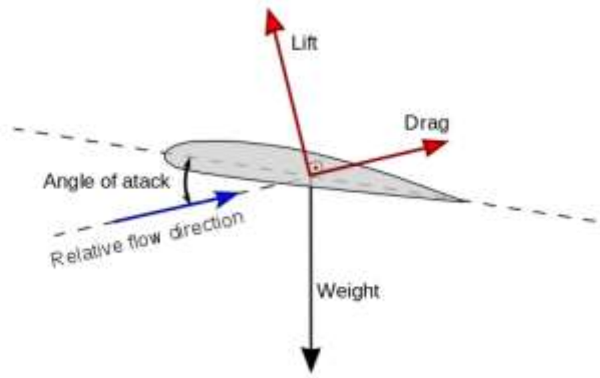

Fig – 1: Direction of Lift and Drag

DESIGN OF AIRFOIL:

Airfoil is an aerodynamic shape which offers the best possible lift for the least amount of drag.

In the airfoil profile, the forward point is called the leading edge and the rearward point is called the trailing edge. The straight line connecting the leading and trailing edges is called the chord line of the airfoil. The distance from the leading edge to the trailing edge measured along the chord line is designated as a chord. The mean camber line is the locus of points midway between the lower surface and upper surface when measured normal to the mean camber line itself. The camber is the maximum distance between the mean camber line and the chord line, measured normal to the chord line. The thickness is the distance between the upper and lower surfaces also measured normal to the chord line. The shape of the airfoil at the leading edge is usually circular, with a leading-edge radius of 0.02c, where c is the chord length. The upper and lower surfaces are also known as suction and pressure surfaces respectively.

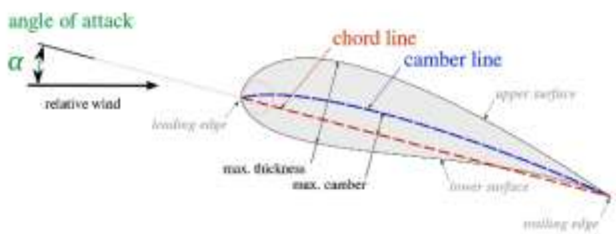

Fig – 2: Structure of an Airfoil

Airfoils are mainly classified based on their shapes and profiles. They are categorized into:

- Symmetric airfoils: Where the upper and lower surfaces of the airfoils are identical to each other.
- Cambered (non-symmetric) airfoils: The upper and lower surfaces are not identical. This is done in order to increase the lift on the aircraft.

The diagram of Symmetric as well as Cambered airfoils are given below:

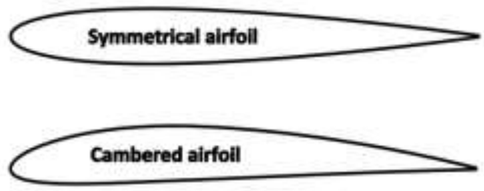

Fig – 3: Symmetric and Cambered airfoil

An airfoil is designed in Solidworks 2016. The overview is given below:

Fig – 4: Front airfoil

Fig – 5: Rear airfoil

DESIGN AND SIMULATION OF FRONT WINGS:

Front wings assembly is designed in Solidworks 2016 software, the isometric view of the design is given below:

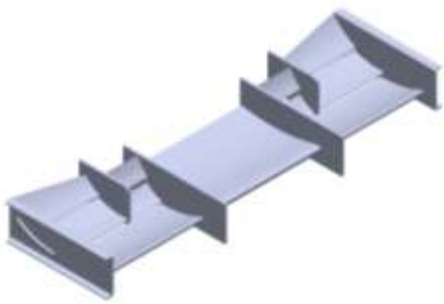

Fig – 6: Front wings assembly

Now, analysis of the wings is very important to understand whether the design of the front wings that is done, if it is able to reduce drag as well as increase downforce or not.

The initial conditions of the simulation are:

Static Pressure: 101325 Pa
Temperature: 293.20 K
Velocity in X direction: 0 m/s
Velocity in Y direction: 0 m/s
Velocity in Z direction: -100.000 m/s

Table – 1: Initial Conditions

The procedure of simulation using Solidworks 2016 is discussed below:

1. Open Flow simulation and select wizard.
2. Click next and select SI units.
3. Select analysis type External and check Consider closed cavities and provide reference axis on which the air is flowing towards the wing.
4. Add air in Gases and provide velocity of the air 100m/s.
5. Adjust computational domain and apply global goals (Force x, Force y, Force z)
6. Run the analysis. Apply goal plots after model is prepared in the solver and wait for the results.
7. Select all the faces in Flow Trajectories and apply Sphere in appearance and click ok and play animation for proper observation.
8. Insert Surface plots and select all the faces and apply Contours and select Pressure / Temperature / Velocity / Density (Fluid).
9. Generate Report.

The result of the simulation is given below:

Name	Value (N)	Progress
GG Force (X)	1.072 N	31 %
GG Force (Y)	-1646.609 N	100 %
GG Force (Z)	-598.120 N	100 %

Table – 2: Force results

This GG Force (Y) is nothing but the downforce that is produced by the wings that is 1646.609 N which is equal to

167.85 kg which seems to be good. The negative sign implies that the force is acting towards the ground.

This GG Force (Z) is nothing but the drag that is produced by the wings that is 598.12 N which is equal to 60.97 kg which is not too much. Here, the negative sign represents that the force is acting opposite the direction of the moving car.

Parameters	Minimum	Maximum
Density (Fluid) [kg/m^3]	1.13	1.29
Pressure [Pa]	95174.57	110162.15
Temperature [K]	290.22	298.16
Velocity [m/s]	0	126.341
Velocity (X) [m/s]	-33.890	30.702
Velocity (Y) [m/s]	-46.920	70.225
Velocity (Z) [m/s]	-126.339	39.547
Mach Number	0	0.37
Shear Stress [Pa]	0	36.93

Table – 3: Simulation results

The pressure that is developed during the air flow is given below:

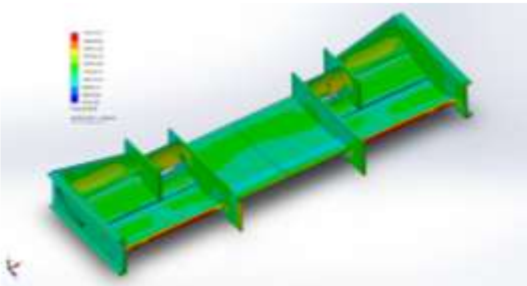

Fig – 7: Pressure Analysis

Minimum pressure developed in the front wings is 95174.57 Pa and maximum value is 110162.15 Pa. Pressure developed at the bottom surfaces is different from the top surfaces.

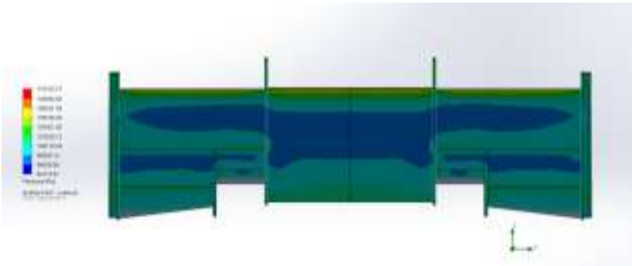

Fig – 8: Pressure at bottom surfaces

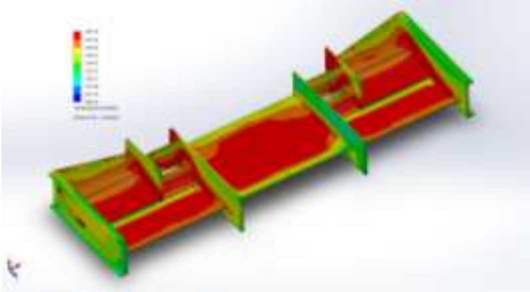

Fig – 9: Temperature (Fluid) Analysis

Minimum temperature developed in the wings is 290.22 K and maximum value is 298.16 K. Temperature developed at the bottom surfaces is different from the top surfaces.

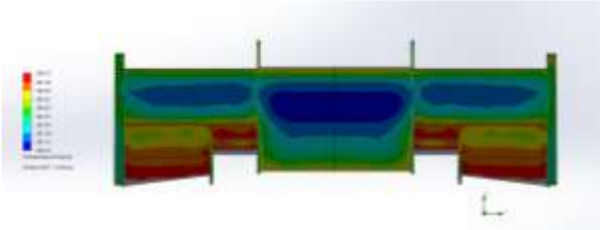

Fig – 10: Temperature (Fluid) at bottom surfaces

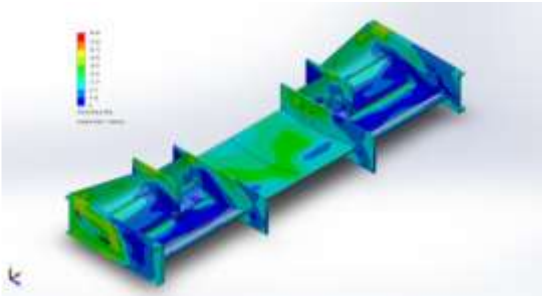

Fig – 11: Shear Stress Analysis

Maximum shear stress developed in the front wing is 36.93 Pa. Shear stress developed at the bottom surfaces is different from the top surfaces.

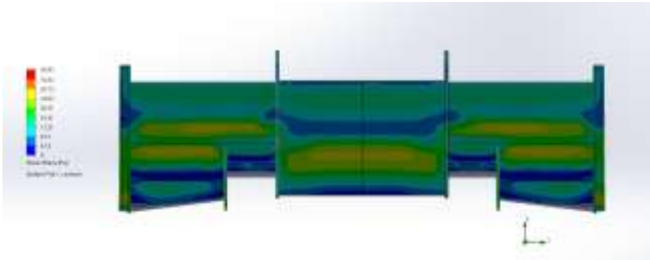

Fig – 12: Shear Stress at bottom surfaces

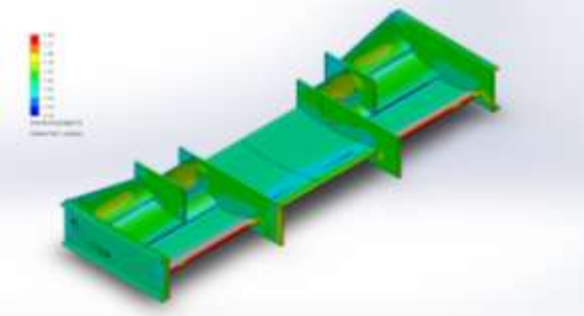

Fig – 13: Density (Fluid) Analysis

Minimum Density of fluid developed in the wings is 1.13 kg/m^3 and maximum value is 1.29 kg/m^3. Density developed at the bottom surfaces is different from the top surfaces.

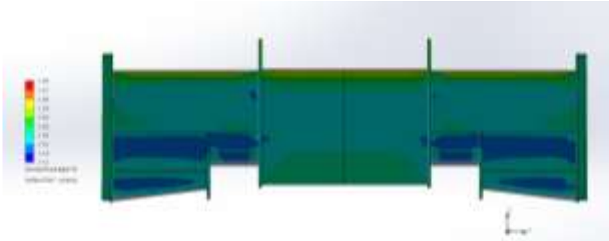

Fig – 14: Density (Fluid) at bottom surfaces

Now, flow trajectory simulation is very important in the case of front wings analysis. It shows proper behavior of the airfoil when it comes into contact with air.

The flow trajectory simulation is shown below:

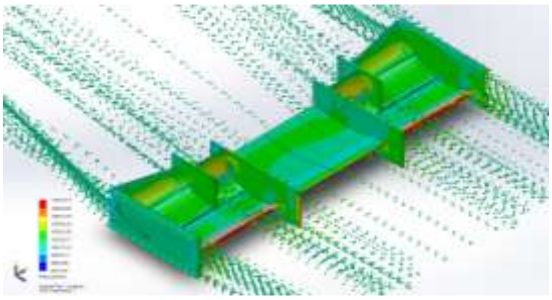

Fig – 15: Flow Simulation of front wing

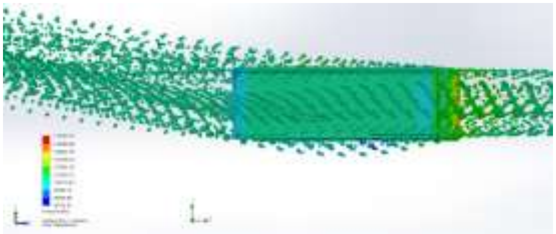

Fig – 16: Side view of Flow Simulation

DESIGN AND SIMULATION OF REAR WINGS:

Rear wings assembly is designed in Solidworks 2016 software, the isometric view of the design is given below:

Fig – 17: Rear wings assembly

Now, analysis of the wings is very important to understand whether the design of the front wings that is done, if it is able to reduce drag as well as increase downforce or not.

The initial conditions of the simulation are:

Static Pressure: 101325.00 Pa
Temperature: 293.20 K
Velocity in X direction: 100.000 m/s
Velocity in Y direction: 0 m/s
Velocity in Z direction: 0 m/s

Table – 4: Initial Conditions

The procedure of simulation using Solidworks 2016 is discussed below:

1. Open Flow simulation and select wizard.
2. Click next and select SI units.
3. Select analysis type External and check Consider closed cavities and provide reference axis on which the air is flowing towards the wing.
4. Add air in Gases and provide velocity of the air 100m/s.
5. Adjust computational domain and apply global goals (Force x, Force y, Force z)

6. Run the analysis. Apply goal plots after model is prepared in the solver and wait for the results.

7. Select all the faces in Flow Trajectories and apply Sphere in appearance and click ok and play animation for proper observation.

8. Insert Surface plots and select all the faces and apply Contours and select Pressure / Temperature / Velocity / Density (Fluid).

9. Generate Report.

The result of the simulation is given below:

Name	Value	Progress
GG Force (X) 1	384.298 N	100
GG Force (Y) 1	-1271.679 N	100
GG Force (Z) 1	-2.076 N	34

Table – 5: Force results

This GG Force (Y) is nothing but the downforce that is produced by the wings that is 1271.679 N which is equal to 129.63 kg which seems to be good. The negative sign implies that the force is acting towards the ground.

This GG Force (X) is nothing but the drag that is produced by the wings that is 384.298 N which is equal to 39.174 kg which is not too much.

Parameters	Minimum	Maximum
Density (Fluid) [kg/m^3]	1.13	1.36
Pressure [Pa]	95260.60	115445.80
Temperature [K]	289.85	298.10
Velocity [m/s]	0	129.880
Velocity (X) [m/s]	-13.747	129.486
Velocity (Y) [m/s]	-58.790	73.289
Velocity (Z) [m/s]	-52.340	51.108
Mach Number	0	0.38
Shear Stress [Pa]	0	31.38

Table – 6: Simulation results

The pressure that is developed during the air flow is given below:

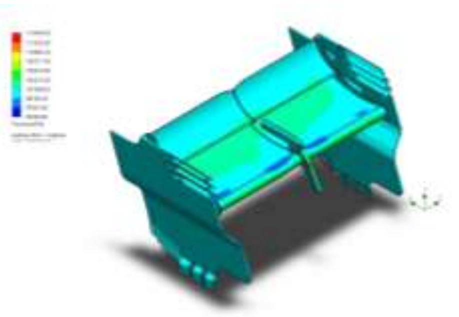

Fig – 18: Pressure Analysis

Minimum pressure developed in the rear wings is 95260.60 Pa and maximum value is 115445.80 Pa. Pressure developed at the bottom surfaces is different from the top surfaces.

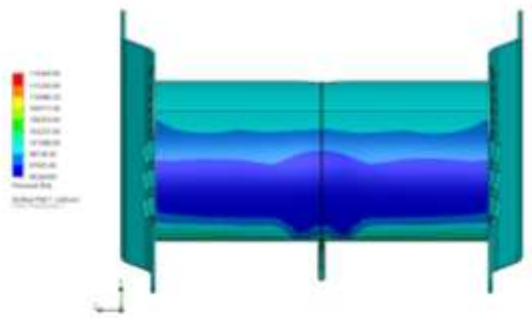

Fig – 19: Pressure at bottom surfaces

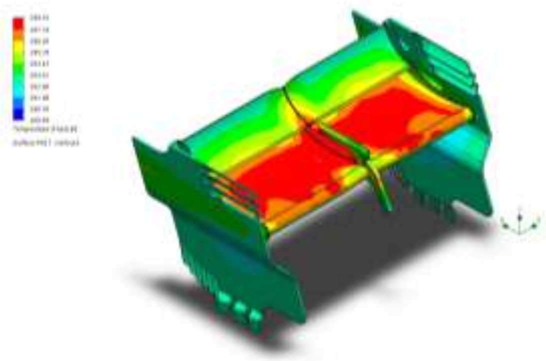

Fig – 20: Temperature (Fluid) Analysis

Minimum temperature developed in the wings is 289.85 K and maximum value is 298.10 K. Temperature developed at the bottom surfaces is different from the top surfaces.

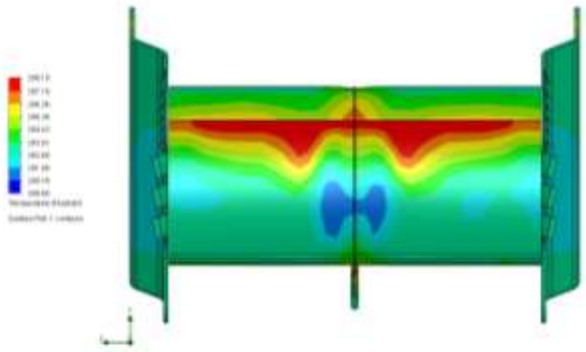

Fig – 21: Temperature (Fluid) Analysis

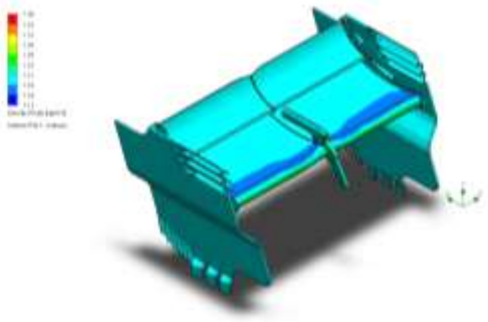

Fig – 22: Density (Fluid) Analysis

Minimum Density of fluid developed in the wings is 1.13 kg/m^3 and maximum value is 1.36 kg/m^3. Density developed at the bottom surfaces is different from the top surfaces.

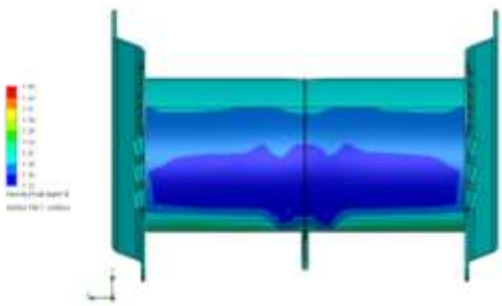

Fig – 23: Density (Fluid) at bottom surfaces

Now, flow trajectory simulation is very important in the case of front wings analysis. It shows proper behavior of the airfoil when it comes into contact with air.

The flow trajectory simulation is shown below:

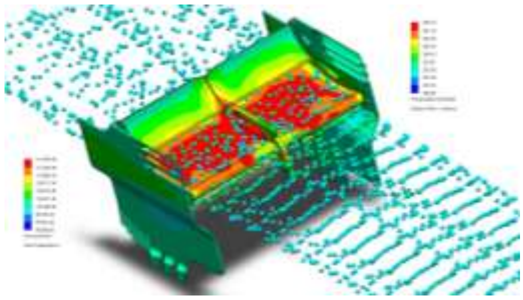

Fig – 24: Flow Simulation of rear wing

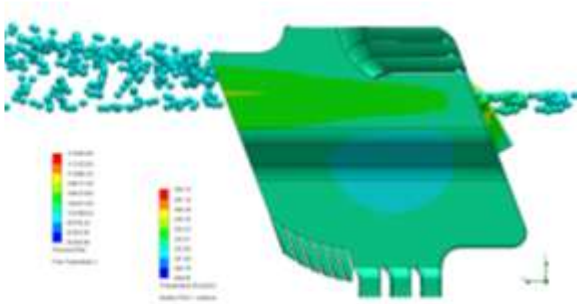

Fig – 25: Side view of Flow Simulation

CONCLUSION:

The first part of the paper discusses about the concept of aerodynamics and the concept of drag, lift, downforce etc. and the second part consists of the design and simulation of front and rear wing of a formula one racing car using Solidworks 2016 software and the analysis procedure which is verified using Computational Fluid Dynamics (CFD) analysis.