International Journal of Latest Trends in Engineering and Technology
Vol.(16)Issue(4), pp.032-038
DOI: http://dx.doi.org/10.21172/1.164.05
e-ISSN:2278-621X

CFD ANALYSIS OF FORMULA 1 RACE CAR

Naveen Raj. L[1] , Dr. Shridhar Kurse[2]

Abstract- In this paper, a study on the dynamics and airflow effects on the Formula 1 race car is presented. This study Integrates Computational Fluid Dynamic analysis and simulation to reduce the drag and to maximize downforce during high-speed travel of the race car. By using the ANSYS software, the simulation employs efficient meshing techniques and realistic loading conditions to understand downforce on the front and rear wing portions of the race car as well as the drag force created by the exteriors of the car body. Flow of air around the car body under the high velocity of the air are illustrated. The direct angle of attack and geometry modifications on the outer body of the car is performed to enhance the downforce and lessen drag for maximum control and stability during operation. The tools provided in the ANSYS proved to be an efficient means of modifying and correcting geometry.
Key Words: CFD, Formula 1, F1, Aerodynamics, Fluid Flow, Race Car.

I. INTRODUCTION

The evolving automotive technology necessitates an increasing complexity associated with its functionalities and safety. An increase in speed leads to an increase in vibrations and greater risk of lift. To guard the cabin system, in addition to enhancing the performance of the car, automotive engineers and manufacturers need to consider the aerodynamics of the car. Simulation acts as a tool to help determine the behavior of the system in response to the airflow around/through it. Using this information, several refinements to the car design can be made in an iterative process.

A **Formula One car** is a single-seat, open cockpit(there is no top closing), open-wheel racing car(the wheel is not covered) with front and rear wings, and an engine positioned behind the driver for better C.G, and used at Formula 1 racing events. The rules and regulations of the cars are differ events to event and specifically that cars should be built by the racing teams themselves. The modern-day Formula 1 cars are constructed from carbon fiber composites and similar ultra-lightweight materials. The minimum weight permissible is 740 kgs including the driver but no fuel included. Cars are weighed with fitted dry weather tyres. According to the 2014 F1 season. The advantage of using weights is to maintain ideal weight distribution and it can be placed anywhere in the car. This can help lower the car's centre of gravity to improve stability and also allows the team to tune the weight distribution of the car to suit individual circuits.

II. METHODS AND MATERIAL

The computational approach is used to analyze the stability of the F1 race car model. In this study, the CFD (Fluent) is used (i.e. ANSYS) as a tool to simulate the aerodynamic characteristics of objects. In this simulation, the domain is assumed to be 3D geometry.

The basic procedural steps are given below:

1. Literature survey.
2. CAD modeling of formula 1 car.
3. Creating the geometric walls and meshing.
4. Input the boundary conditions.
5. Solving the Problem and obtain results

[1]*Department of Automobile Engineering,* New Horizon College of Engineering, Bengaluru, Karnataka, India
[2]*Department of Automobile Engineering ,* New Horizon College of Engineering, Bengaluru, Karnataka, India

A. AERODYNAMICS OF AUTOMOBILES-

Automobiles started using aerodynamic body shapes in the early part of their history. As engines became more powerful and cars became faster, automobile engineers realized that at the same time wind resistance reduced their speed. The first cars to adopt improved aerodynamics or the streamlining, were racing cars and those breaking the race records.
Regarding the aerodynamics of a racing car, Dr. Joe David, professor of mechanical and aerospace engineering known as "Mr. Stock Car" at North Carolina State University, said, "Most of the horsepower generated by a racing engine is eaten up by the high-pressure air pushing the front of the car and the low-pressure air dragging at the car from behind."

B. AERODYNAMIC CHARACTERISTICS -

1.2.1 DRAG FORCE

The combination of the effects of pressure and wall shear forces creates the drag in the airflow direction. This force is because of the consolidate impact of the wall shear pressure and weight powers is characterized by Equation:

$$C_d = \frac{F_d}{\rho V^2 \frac{A}{2}} \quad (1)$$

[source: National Aeronautics and Space Administration]

Where, C_d is the drag coefficient, ρ is the density of air, V is the flow velocity relative to an object, F_d is the drag force and A is the cross-sectional area.

1.2.2 LIFT FORCE

It is the vertical segment of the resultant force brought about by the pressure approximately on the vehicle. The aerodynamic lift and pitching moment are undesirable effects. The aerodynamic lift tends to reduce the pressure between the tire and the ground. This causes the loss of steering on the front axle and loss of traction on the rear axle. Pitching causes rear wheel lift off the ground and or reduces available traction. It is the rocking chair or pivoting activity about the transverse hub through the vehicle corresponding to the ground. Because of pitching, the front suspension moves out of stage with the back bringing about shaking impact in a vehicle. The coefficient of lift equation is given by:

$$C_l = \frac{2F_l}{\rho V^2 A} \quad (2)$$

[source: National Aeronautics and Space Administration]

Where C_l is the coefficient of lift, F_l is Lift force, A is the cross-sectional area, V is the flow velocity relative to object and ρ is the density of air.

1.2.3 SIDE FORCE

The improper wheel balance creates the centrifugal force that acts on the vehicle during turning and produces a side thrust force. To sustain that force, the plane of the wheel makes some angle with the direction of the motion of the vehicle. This is achieved by the direction of the tire which is flexible. The angled form during taking turns to sustain the side thrust is known as slip angle and the force produce to counteracts the side thrust is called a cornering force.

1.2.4 ROLLING MOVEMENT

It is the movement of the vehicle about its longitudinal axis generated due to centrifugal force act during cornering. During cornering, a turning couple is produced about the longitudinal axis of the vehicle owing to centrifugal force acting at the centre of gravity and forces acting at point of contact of road and tire patch. This results in a motion known as rolling. A combination of rolling and pitching is called diagonal pitch.

C. LITERATURE REVIEW

Information regarding the Aerodynamics of Formula 1 was obtained by the Journal paper- [2].
Information regarding the CFD analysis and simulation was obtained by the Journal papers- [1], [3] and [4].
Information regarding the Drag was obtained by the Journal paper- [5].

III. DIMENSIONS

According to the technical regulations, an F1 car should have the following dimensions:

- The width of the car must not exceed 2000 mm.
- The length, height, and shape of the car are effectively governed by Different parameters.
- Certain pieces of bodywork, such as front wing endplates, must be at least 10mm thick. This is to prevent tyre damage.
- No part of the car can be more than 950 mm in height.
- The car's survival cell structure, or the monocoque, designed to protect the driver in the event of an accident, must extend at least 300mm beyond the driver's feet.

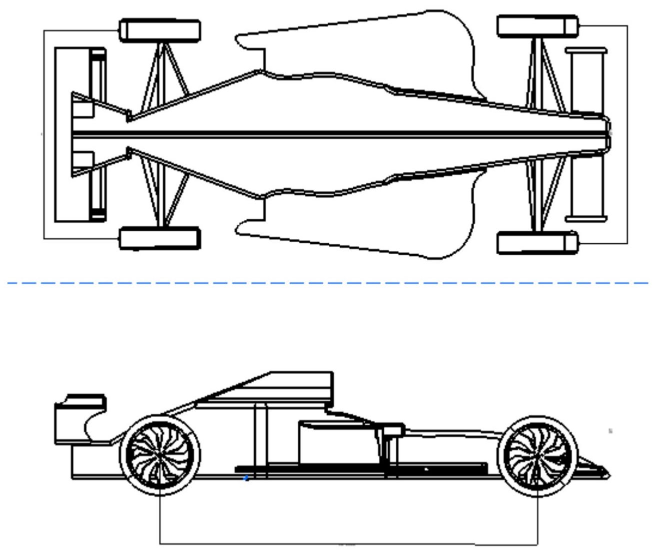

Fig 1 Draft of Top and Side View

We have maintained the following dimensions:
WHEEL BASE=3372mm
FRONT TRACK=1950mm
REAR TRACK=1860mm

IV. CAD MODELLING

Fig 2 ISOMETRIC VIEW

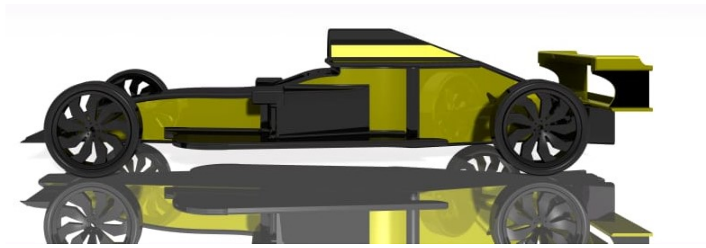

Fig 3 SIDE VIEW

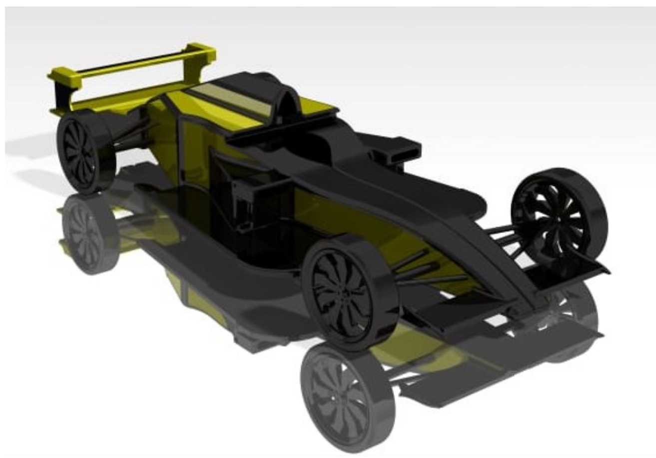

Fig 4: TRIMETRIC VIEW

Fig 5: Front View

V. METHODS AND RESULTS OF ANALYSIS DONE ON ANSYS

2.1 Formula 1 car CAED file is imported and a cuboidal closed system is created around the car.

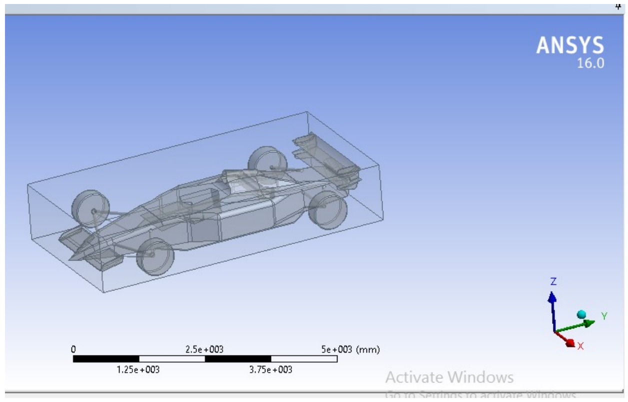

Fig 6 Cuboidal Closed System

2.2 Medium size meshing is done. The number of nodes formed was 296544 and the number of Tetrahedral elements formed was 1650524.

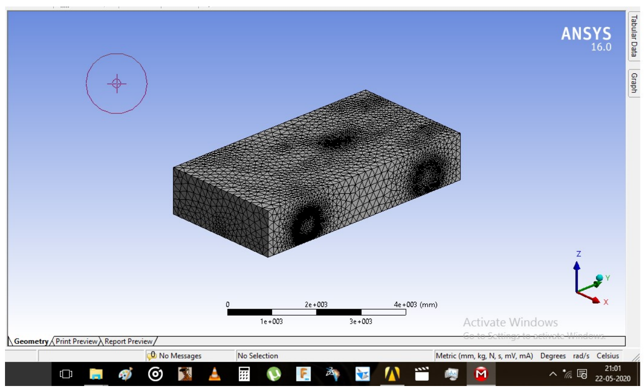

Fig 7 Meshing of the Car

2.3 In the Boundary conditions, attack is given an inlet velocity of 120m/s and escape is given as air outlet.

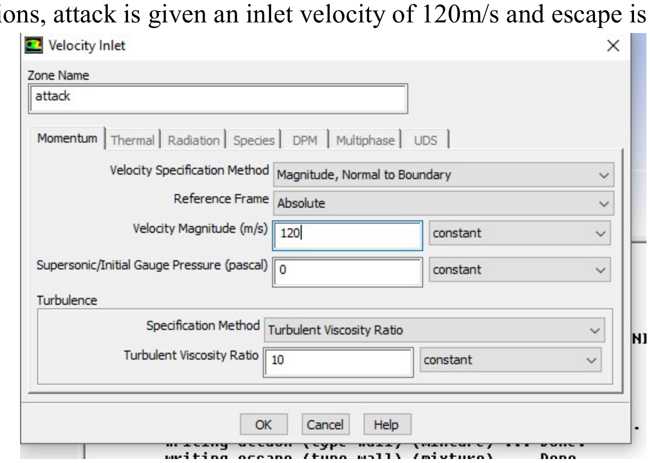

Fig 8 Velocity Input

2.4 In the monitor section, Drag and lift options are selected and inputs are given.

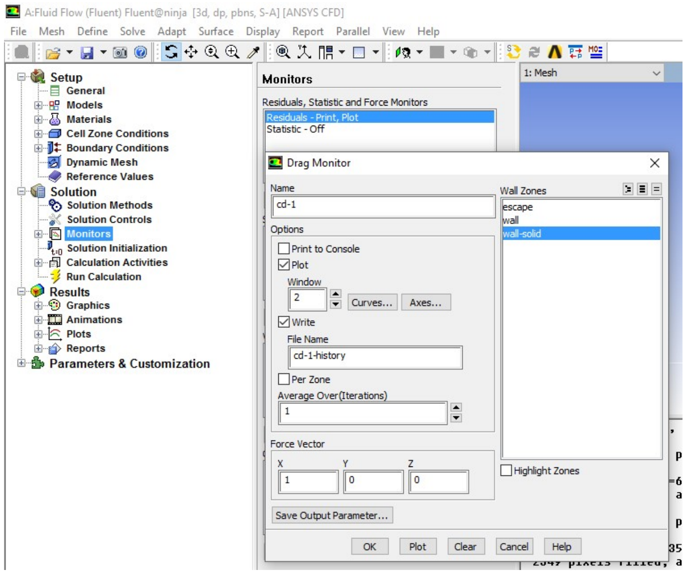

Fig 9 Drag Monitor

2.5 Calculation is run for 200 iterations

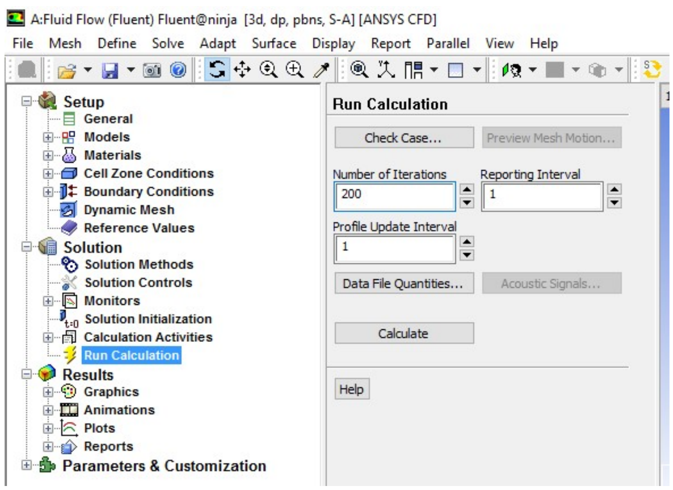

Fig 10 “Run Calculation” Tab

2.6 After 200 iterations, we get the straight horizontal line and represent the coefficient of drag as 0.85

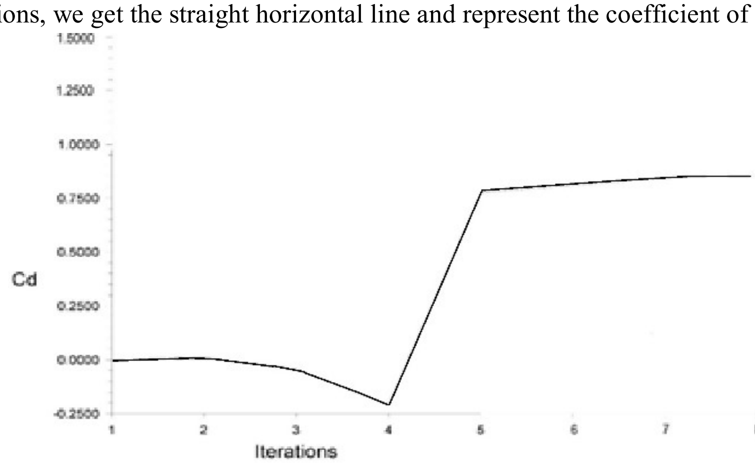

Fig 11 Graph of Coefficient of Drag

2.7 Pressure streamline flow is checked around the wheels

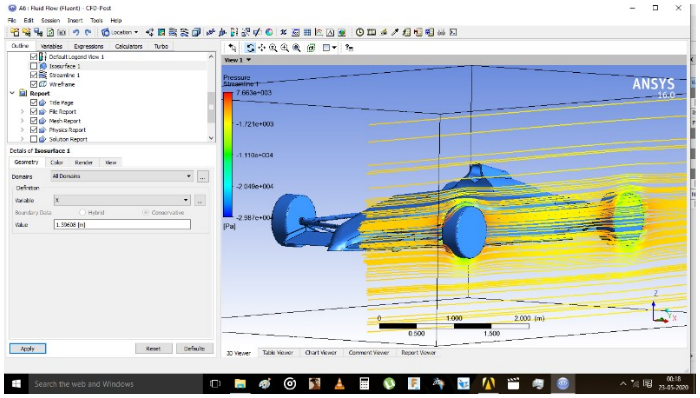

Fig 12 Pressure streamline flow around Wheels

IV.CONCLUSION

From the above study we arrive at the following:

1. Aerodynamics plays an important role in the design of any vehicle.
2. Here, in this paper, an attempt was made to create an F1 racing car model and to analyze the stability of the car including the drag produced during high performance.
3. Using CATIA V5 software, we generated an F1 racing car model.
4. Using Ansys 16.0 workbench we have analyzed the stability of the vehicle.
5. The coefficient of drag of formula 1 car is between 0.7 to 1.1, In our study, at the velocity of 120m/s, the coefficient of drag was found to be 0.85 of the modeled Formula 1 car.

REFERENCES

[1] Shubham Borole, Omkar Bhasale, Shubham Patil, Akshay Khot and Prof. Vivekanand Navadagi. 2016. “Analysis of Aerodynamic Characteristics In Front Wing of F1 Car Using CFD”, International Journal for Scientific Research & Development, (June 2016), 6th ed. ISSN NO:2395-1052

[2] R. K. Pethar, S. G. Kolgiri, and S. S. Ragit. 2014. “Study of Front-Body of Formula-One Car For Aerodynamics using CFD”, International Journal of Application or Innovation in Engineering & Management. (March 2014), Volume 3, Issue 3. ISSN: 2319-4847

[3] Senan Thabet and Thabit H Thabit. 2018. “CFD Simulation of the Air Flow around a car Model”, International Journal of Scientific and Research Publication, (July 2018), Volume 8, Issue 7. ISSN: 2250-3153

[4] Manoj Kumar D. Birajdar, Suresh Choudhary and Prof. Vivek Mane. 2017. “CFD Analysis of an Automobile to Improve the Aerodynamics”, International Journal Of Advance Scientific Research And Engineering Trends. (September 2017), Volume 2, Issue 2. ISSN: 2456-0774

[5] A. Muthuvel,and N. Prakash. 2014. “Numerical Simulation Of Drag Reduction In Formula One Cars”, J. Godwin John International Journal of Engineering Research and Applications. (29th March 2014), ISSN : 2248-9622