DESIGN AND FLUID FLOW ANALYSIS OF F1 RACE CAR

Banavath Venkatesh[1], P. Chandra kumar[2]

[1] M.Tech student, Department of Mechanical Engineering, Vidya Jyothi Institute of Technology, Hyderabad, TS.
[2] Assistant Professor, Department of Mechanical Engineering, Vidya Jyothi Institute of Technology, Hyderabad, TS.

---***---

Abstract :- *The main criteria of this paper is to presents the method of modeling and design of the racing car by using the modern software and to analysis of the external factor that on the body by using CFD techniques. The work is based on elements of aerodynamics, vehicle dynamics & impact of external-influences on it. The model confirmation with numerical calculations was showed in this research after a detailed analysis of the obtained result with CFD, it was conducted that the model of the car with geometric simplification and parameters meets the criteria of accuracy and this could be useful further course of analysis. The designing of product by using SOLIDWORKS and analysis were done by CFD FLUENT.*

Key words: Aerodynamics, Car body, solid works CFD analysis, Ansys.

1. INTODUCTION

CFD is broadly used as an analysis software in this paper F1 car drag force & down force were analyzed at high velocity. CFD used to analyze by numerical equations such as governing equations of Fluid dynamics throughout required region. It allow to solve complex problem and gives an accurate solution without disturbance of structure of the body. In recent all industries were using CFD software to get an exact and accuracy of the results and also it reduces the time as well investment. Mostly this software is used at production Industries. CFD is mostly used at flowing and moving objects such as flow of water, fluids and aerodynamic effect at wings or hood at particular part of the vehicle. CFD allows of mapping airflow through the engine, and even within the car to estimate the behavior of thermal comfort systems and the efficiency of cooling systems. One of the most significant aspects of a F1 racing car design is aerodynamics. Making down power, to hold the auto to the ground to enhance cornering; and limiting drag, which backs the auto off are two essential concerns when outlining the auto. Present day dashing auto groups utilize costly breeze burrows and computational liquid elements frameworks to break down the viability of a streamlined outline for an auto.

F1 cars will have small winglets before the back wing, which clean up complex air flow in order to maximize down force. The scope of this project includes. These fundamental principles can be uttered in terms of mathematical equations, which are the most general form are usually partial differential equations. The body is developed by steel tubing with least dimensional and quality. The ISIE association manual confines the vehicle weight, shape, size, and measurement. CFD enables specialists to look at the wind stream over a car or a specific part, for example, a wing or hood and see the streamlined impact and, takes care of complex issues without losing their respectability because of simplicity of the product. The fundamental points of interest of utilizing CFD programming is that the outcomes are gotten without development of the required model and this is imperative since it can lessen the expense in building the F1 autos. The legitimacy of results is the most imperative thing that we have to worry about while utilizing the product simulation. Therefore specific parameters and conditions while analyzing the data need to be valid.

Among different parts that add to the fluctuating levels of down force it is simply the front wing that loan to hypothetical aerodynamic examination strategy and procedures for configuration utilizing the CFD programming. The job of CFD in building figure has turned out to be strong to the point that today it might be seen as another third measurement of liquid elements, the other two being unadulterated experimentations and unadulterated hypothesis. Vehicle aerodynamics is a broad term assimilate the field that describes the forces acting on an object when it passes through the fluid. Aerodynamicists study this natural phenomenon to try & minimize forces that inhibit motion and in some cases, develop these forces and use them to improve performance and safety.

2. LITREATURE OF REVIEW

The staging of the front wing of the Formula One car is significantly affected by the existence of ground. The nearby ground helps to develop a large net down force, also known as the negative lift when the flow is simulated with the ground effect (Kieffera et al., 2005).

After studying the analysis of the model it can be inferred that the vertical plate deflects the flow of air, thus reducing the drag force and also further prevents the super pressure the wheel to extent to multi-element air foil surfaces thereby increasing the down force. Aniruddha Patil [1] et al. The throttle bore parameters were designed using flow equations and was cast.

The flow through the throttle frames each with different shaft profiles were simulated and compared experimentally. Data obtained from both showed the rectangular shaft profile to have less pressure drop as compared to the circular one. The throttle body with rectangular shaft profile showed better flow characteristics with minimal backflow and reduced turbulence downstream of the butterfly valve thus proving to be the better choice.

An IC engine equipped with such a throttle body will have better breathing capacity which would mean an improvement its Performance, fuel economy and emissions M. Balaji[2] et al. CFD results for many managements of the rear slant angle of the car Body are granted after the geometries are designed & then developed.

Comparison & examination by plotting a graph of the lift & drag coefficients obtained for the numerous managements is done to determine the optimum rear slant angle giving minimum drag coefficient. Saurabh Banga [3] et al. Flow perception, hot wire, and LDA estimations have been performed around body for two diverse back inclination points: 25° and 35°.

These two inclination points section a basic shakiness when the flow disconnects from the inclined surface. LDA estimations incorporate every one of the three parts of mean and RMS speed and in addition most second and third request minutes. The estimations obviously demonstrate the distinctions in stream connection and distribution for the two diverse inclination points considered. The quantitative data given by these estimations ought to demonstrate priceless for creating, testing, and approving computer models of the streamlined features of vehicular wake regions H. Lienhart [4] et al.

From the outcomes at the end of this design and fabrication, there are limited highlights that can be measured as meeting the objectives & gaining values. The aim is to found alternative way to manufacture formula Varsity race car body. The required design method are performed, which shows that alternative leads to better product in some perspectives & these all are supported by figures. S. M. Sapuan [5] et al.

The Results of the CFD Analysis of the Front Wing with and without dimple is made at various Velocities and results are compared Raj Kamal M.D [6] et al. The aerodynamic of racing cars can be better by CFD tools. The tool results shows the accurate within a short period of time and the results are not generally not grid and not model independent.

However, once these limitations are properly understood, these tools may be used to reduce the design cycle that must also rely on wind tunnel & track testing. Without validation, results of simulations are indeed of no value. Shubham Borole [7] et al. It can be detected how vehicle dynamics is related to aerodynamics and the way both of them affect each other. In order to exploit the car's performance and improve its cornering capability extra down force was achieved with the help of wings. Dynamic behavior of the car during its maneuvering was not compromised at any cost. Also, keeping safety and comfort as our priority, a trade-off was made between them.

By attaining the required amount of down force from the front & the rear wings, the maximum amount of lateral & longitudinal force which can be generated by the tyres was attained, thereby harnessing its full potential. The tyres used, have the capability to generate a lateral acceleration as high as 2.7g and hence it needs to be made sure that the vehicle does not topple due to the centrifugal force acting on it. As per the calculations, these wings generate an appropriate amount of down force to keep all 4 wheels intact to the ground. Nisha Rastogi [8] et al.

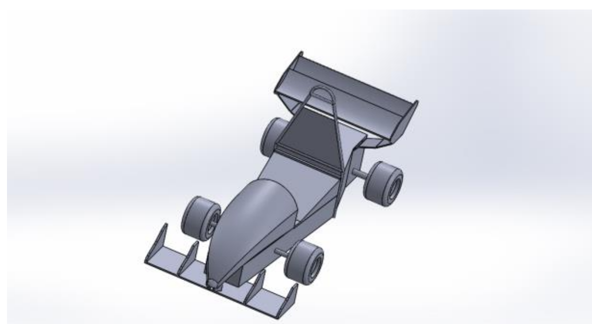

Fig-1: Isometric view of the formula one car

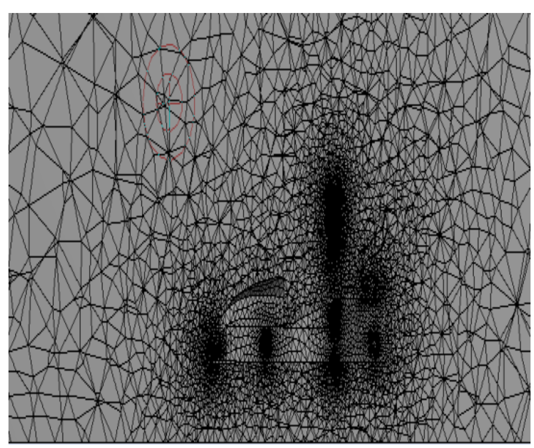

Fig-2: Mesh area of the formula one car.

3. THEORETICAL DESIGN AND CALCULATION

3.1 EXPERIMENTAL STUDY

Estimation of Drag Coefficient and Drag Force on the Base of Field Study

X=distance travelled after switching off the engine
M=mass of vehicle in kg.
V=velocity at which the engine was switched off (In m/s)
Cd =coefficient of drag
p=density of air
A= projected area

The force F restricts the movement of the vehicle For skew of straightforwardness we accept the distinction of F is certain in course of velocity V. Moving obstruction and slope opposition for a given vehicle and inclination individually, are consistent.

Rolling resistance + Gradient resistance =b

Any object standing in the path of a substance flow exerts different values of drag force depending on the square of the velocity the shape of the object and the related flow direction surrounding the object.

$$F_d = \frac{1}{2}\rho A v^2 C_d$$

$$C_d = \frac{2}{\rho A}k \qquad k = \frac{\rho A}{2}C_d \qquad \rho = \frac{2k}{AC_d} \qquad F_d = kv^2$$

ρ – Substance density
A – Cross section area of object facing flow
v – velocity of substance related to object
C_d – Drag coefficient
F_d – Drag force

3.2 SAMPLE CALCULATION OF DRAG FORCE

Drag force-

For front wings:

Length of wing = 1m

Thickness of wings = 10 mm = 0.01m

Width of wing = 20cm = 0.2m

Density of air = 1.29 kg/m^3

Velocity of car = 25m/sec

Surface area = 1*0.2cos (80) + 1*0.01cos (10)

Drag coefficient = 0.85

Drag force = 0.5*(ro)*v^2*S*c

= 0.5*1.29*40^2*0.044*682

= 30.877N

For spoilers:

Length of spoilers = 0.57*2 m

Width of spoilers = 0.30*2 m

Thickness of spoilers = 10mm = 0.01m

Drag force = 0.5*1.29*40^2*0.682*0.0353

= 24.772N

For main body:

Area of body = 0.83m^2

Drag force = 0.5*1.29*40^2*0.682*0.85

= 584.173N

Down force-

Total force on the car = m*a

= m*0.71

= 270*0.71

= 191.7 N

For spoilers:

The spoilers usually contribute about 35% of the down force

The down force acting on spoilers = 0.35*total force

= 0.35*191.7

= 67.09.N

For wings:

Since wings contribute for the remaining down force

Down force on wings = 0.65*total down force

= 0.65*191.7

= 124.605N

Similarly for the Remaining Values for 60m/S and 80m/S Velocity Substitute. The Velocity Value in the above equations

4. RESULT AND DISCUSSION

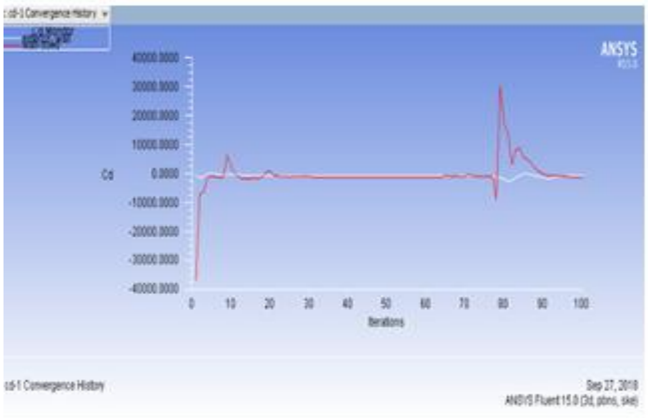

Fig-3: The Graph between Cd Value of External and Internal Flow to the Iteration Value.

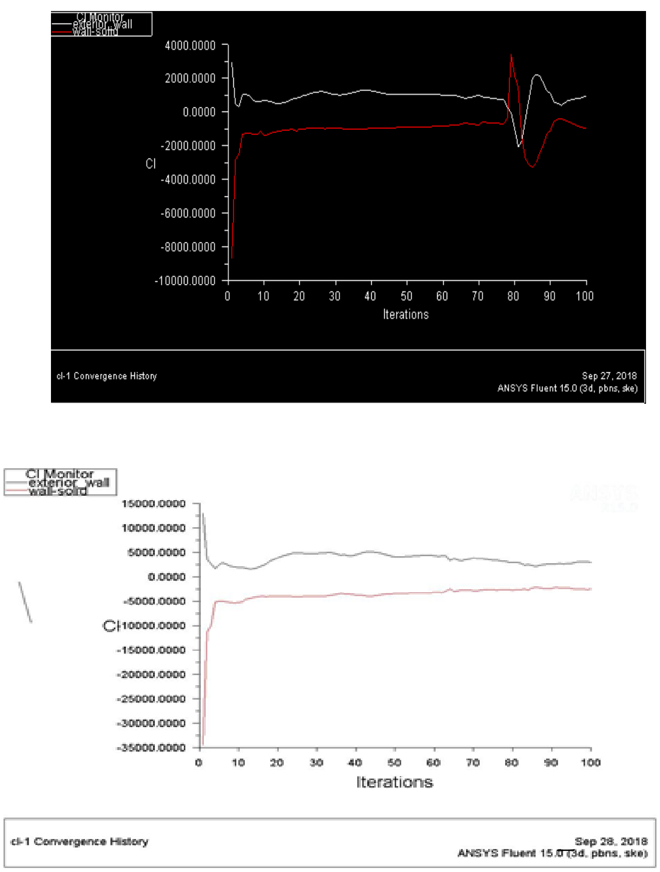

Fig-4: The graph between cl value of external and internal flow to the iteration value

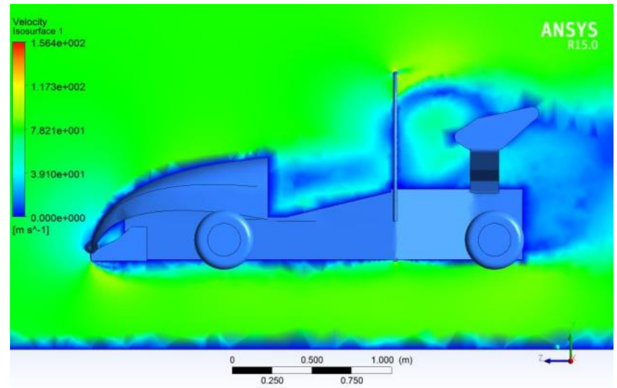

Fig-5: Velocity diagram

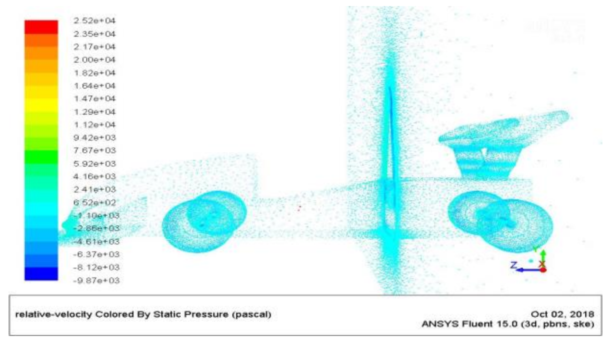

Fig-6: Relative-velocity colored by static pressure

4.1 EXPERIMENTAL DESIGN AND ANALSYS

Table 1. Experimental Design and Analyses

S.NO	VELOCITY m/s	AREA m^2	MESH		DRAG FORCE N	SKIN FRICTION
			NO.OF NODALS	ELEMENTS		
1	40	5.51	336699	1870229	97.27	158.82
2	60	5.51	336699	1870229	56.224	91.795
3	80	5.51	336699	1870229	509.484	831.81

4.2 COMPARISONOF EXPERIMENTAL AND ANALYTICAL

Table 2. Comparison of experimental and analytical

MODEL OF RACING CAR	DRAG COEFFIENT (Cd)	MEAN DRAG FORCE (Fd)
Theoretical calculation	0.746	72.91
Experimental calculation	0.841	221
Difference	0.095	148.09

5. CONCLUSION

The Drag coefficient founded practically and numerically are very similar the result difference value is 0.095.this shows the accuracy of the result found. The drag force are also obtained by the based on the examined values of forces and coefficients obtained from analytical and CFD simulation difference of a mean drag force is 148.09N.

6. REFERANCES

[1] Aniruddha Patil1*, Siddharth Kshirsagar[1] and Tejas Parge1 study of front wing of formula one car using computational fluid dynamics.

[2] M. Balaji* , K. Amal Satheesh, G. Sanjay and Haynes K. job. design of throttle body: a comparative study of different shaft profiles using cfd analysis.

[3] Saurabh Banga1, Md. Zunaid[2]*, Naushad Ahmad Ansari[3], Sagar Sharma[4], Rohit Singh. CFD Simulation of Flow around External Vehicle: Ahmed Body.

[4] H. Lienhart, C. Stoots, S. Becker Flow and Turbulence Structures in the Wake of a Simplified Car Model (Ahmed Model)

[5] S. M. Sapuan*, K. W. Ham, K. M. Ng, C. K. Woo, M. K. A. Ariffin, B. T. H. T. Baharudin, A. A. Faieza, E. E. Supeni and N. A. A. Jalil Design of composite racing car body for student based competition

[6] Raj Kamal M.D, Socrates. S ,Kaliappan Aerodynamic Effects on Formula One Car Using CFD.

[7] Shubham Borole1, Omkar Bhasale, Shubham Patil, Akshay Khot, Prof. Vivekanand Navadagi. Analysis of Aerodynamic Characteristics In Front Wing of F1 Car Using CFD.

[8] Nisha Rastogi, Siddhanth Shetty and B Ashok. Analysing the effects of air flow on a formula prototype vehicle to optimize its performance