# Aerodynamic Design of Front Wing Based on CFD

Jinto Thottan
PG Scholar
Department Of Mechanical Engineering
SRM University, India

Mr. S. Balamurugan
(Asst.Professor (Sr.G),
Department Of Mechanical Engineering
SRM University, India

***Abstract*—Fluid dynamics is a field of science which studies the physical laws governing the flow of fluids under various conditions. Great effort has gone into understanding the governing laws and the nature of fluids themselves, resulting in a complex yet theoretically strong field of research. Computational Fluid Dynamics or CFD as it is popularly known is used to generate flow simulations with the help of computers. CFD involves the solution of the governing laws of fluid dynamics numerically. The complex set of partial differential equations is solved on in geometrical domain divided into small volumes, commonly known as a mesh (or grid).This project deals with aerodynamic developments that result from research in the highly competitive environment of motor sport especially race car are used as a competitive advantage for the team or manufacturer that funded the research. The project concentrated on the wind flow analysis using CFD method, for that ANSYS fluent software is used for analyzing the model and find out appropriate down force. The test is repeated with various inputs and compares the outputs. At last find the optimized down force to reduce the drag while the race car is under certain speed using CAD model.**

***Keywords*— Fluid dynamics; CFD; ANSYS fluent; down force***

## 1. INTRODUCTION

Fluid dynamics is a field of science which studies the physical laws governing the flow of fluids under various conditions. Great effort has gone into understanding the governing laws and the nature of fluids themselves, resulting in a complex yet theoretically strong field of research.

Computational Fluid Dynamics or CFD as it is popularly known is used to generate flow simulations with the help of computers. CFD involves the solution of the governing laws of fluid dynamics numerically. The complex set of partial differential equations is solved on in geometrical domain divided into small volumes, commonly known as a mesh (or grid).

From the 1960s onwards the aerospace industry has integrated CFD techniques into the design, R&D and manufacture of aircraft and jet engines. More recently the methods have been applied to the design of internal combustion engine, combustion chambers of gas turbine and furnaces. Furthermore, motor vehicle manufacturers now routinely predict drag force, under-bonnet air flows and the in car environment with CFD. Increasingly CFD is becoming a vital component in the design of industrial products and processes.

The ultimate aim of development in the CFD field is to provide a capability comparable to other CAE tools such as stress analysis code. The main reason why CFD has lagged behind is the tremendous complexity of the underlying behavior, which precludes a description of fluid flow that is at the same time economical and sufficiently complete. The availability of friendly interfaces has led to a recent upsurge of interest and CFD is poised to make an entry into the wide industrial community in the 1990s..

## 2. OBJECTIVE

This project is focused on the aerodynamics of a race car. The project concentrated on the wind flow analysis using CFD method. ANSYS fluent is used for analyzing the result and find out appropriate down force. The test is repeated with different input and compares the outputs. At last find the optimized down force to reduce the drag while the car is under certain speed and make necessary changes in the design of the front wing.

## 3. FRONT WING

A regular front aerofoil is made as a main plane running the whole width of the car suspended from the nose. Onto this are fitted one or more flaps which are the adjustable parts of the wing. On each end of the main plane there are endplates.

The front wing of a race car operates in ground effect and produces about 25%–30% of the total downforce of the car. The downforce, or aerodynamic grip, works in conjunction with the mechanical grip, to improve the acceleration, braking, and cornering speed of the car.

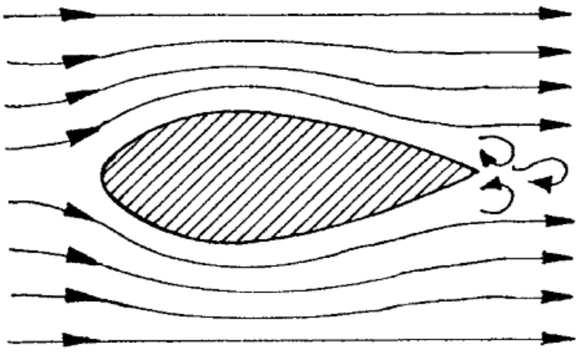
However, it is not only the overall level of downforce that is the important factor. As the car accelerates or brakes, the suspension movement on the car causes the front wing to change height above the ground. This influences the level of downforce produced by the front wing, and in fact that by under tray and diffuser as well. In terms of drivability, the best performing car is a well balanced one.

## 4. AERODYNAMICS

The constant need for better fuel economy, greater vehicle performance, reduction in wind noise level and improved road holding and stability for a vehicle on the move, has prompted vehicle manufacturers to investigate the nature of air resistance or drag for different body shapes under various operating conditions. Aerodynamics is the study of a solid body moving through the atmosphere and the interaction which takes place between the body surfaces and the surrounding air with varying relative speeds and wind direction. Aerodynamic drag is usually insignificant at low vehicle speed but the magnitude of air resistance becomes considerable with rising speed. A vehicle with a high drag resistance tends only marginally to hinder its acceleration but it does inhibit its maximum speed and increases the fuel consumption with increasing speed.

## 5. AERODYNAMIC DRAG

When viscous air flows over and past a solid form, vortices are created at the rear causing the flow to deviate from the smooth streamline flow. Under these conditions the air flow pressure in front of the solid object will be higher than atmospheric pressure while the pressure behind will be lower than that of the atmosphere, consequently the solid body will be dragged in the surface. Where a flat plate placed at right angles to the air movement will experience a drag force in the direction of flow represented by the pulley weight which opposes the movement of the plate.



Aerofoil section

## 6. GOVERNING EQUATIONS

The cornerstone of computational fluid dynamics is the fundamental governing equations of fluid dynamics—the continuity, momentum and energy equations. They are the mathematical statements of three fundamental physical principles upon which all of fluid dynamics is based:

(1) Mass is conserved

(2) F = ma (Newton's second law)

(3) Energy is conserved.

A solid body is rather easy to see and define; on the other hand, a fluid is a 'squishy' substance that is hard to grab hold of. If a solid body is in translational motion, the velocity of each part of the body is the same; on the other hand, if a fluid is in motion the velocity may be different at each location in the fluid.

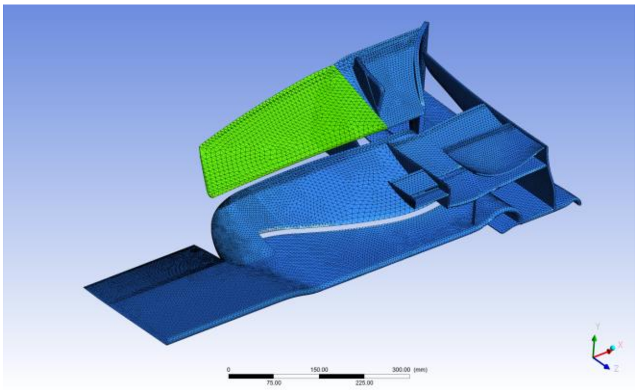
## 7. DATA ABOUT RACE TRACKS

| TRACK | PARAMETERS | DATA | |
|---|---|---|---|
| Monza, Italy | Top speed | 337 | Km/h |
| | Down force | low | - |
| | Wind speed | 2 | m/s |
| Indian gp | Top speed | 321 | Km/h |
| | Down force | Medium | - |
| | Wind speed | 12 | m/s |
| Yas Marina, Abu Dhabi | Top speed | 310 | Km/h |
| | Down force | High | - |
| | Wind speed | 9 | m/s |
| Monza, Italy | Air temperature | 28 | °C |
| | Track temperature | 40 | °C |
| | Humidity | 27 | % |
| Indian gp | Air temperature | 29 | °C |
| | Track temperature | 38 | °C |
| | Humidity | 90 | % |
| Yas Marina, Abu Dhabi | Air temperature | 29 | °C |
| | Track temperature | 34 | °C |
| | Humidity | 58 | % |



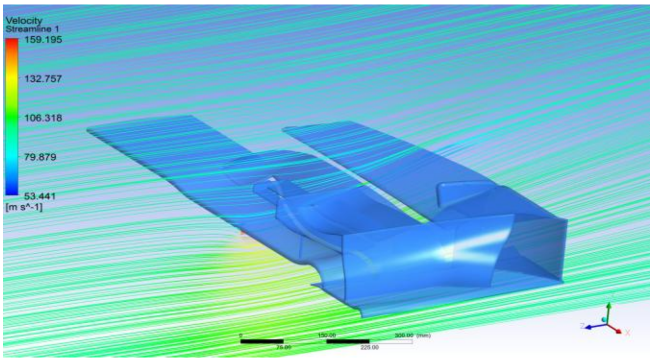
## 8. CFD ANALYSIS ON FRONT WING

Meshing is done using Hyper Mesh and the element used for this model is solid 45 which are also used in ANSYS for analysis.

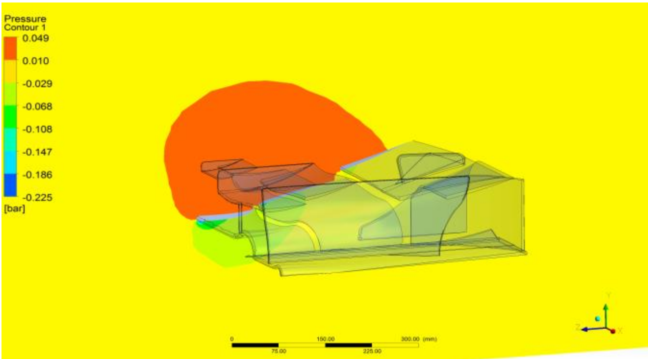


The velocity contour is plotted to show the flow has the prescribed velocity which is given as the input value in ANSYS. The streamline are a family of curves that are instantaneously tangent to the velocity vector of the flow.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 10, October- 2014



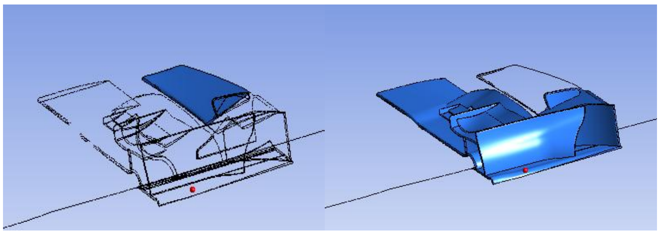

This is the analysis result of the front wing using ANSYS fluent and the pressure contour is plotted to show that the aerofoil has high pressure applied over it.




## 9. CALCULATION OF DOWNFORCE & RESULT VALIDATION

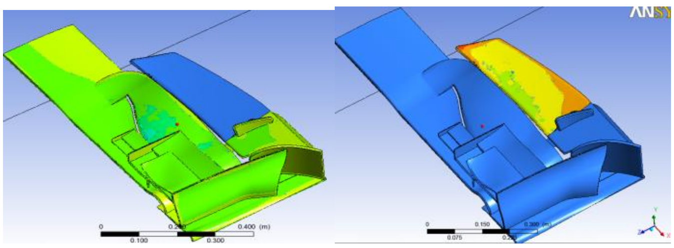
Projected Area of Front Wing

Total area of the front wing = 1.009089 m^2



Lift Acted on the Front Wing

Total lift in y direction = 1413.261 N



COEFFICIENT OF LIFT

L = ½ x ρ x v^2 x A x CL
L = lift force
ρ = air density
v = air velocity
A = area of the wing
CL = coefficient of lift

CL = 0.26093542

DOWNFORCE

DF = ½ x ρ x v^2 x A x α x CL
DF = Down force
ρ = air density
v = air velocity
A = area of the wing
CL = coefficient of lift
α = angle of attack

| Angle | Angle in rad | Down force in N |
|---|---|---|
| 9 | 0.15708028 | 2177.031525 |
| 10 | 0.174533665 | 2418.924202 |



## 10. CONCLUSION

The project discussed about the formula one car and its properties, different parts of the car mainly wings, diffuser etc. The front wing plays a major part of down force which is acted on the vehicle. To increase the down force to an appropriate level and thus reduce the drag that car produce while racing. The aerodynamics of the race car is an important factor which this project deals with. The journals focused on three main aerodynamic components which operate in ground effect: wings, diffusers, and wheels. The front wing of a formula one car was modelled using CAD software's and various views are plotted. By FIA regulations the maximum down force acted over the front wing is about 25% to 35% of total weight of the race car. The down force by above condition is 2369.115 N. Down force obtained by my front wing design at 10 degree angle of attack is 2418.924202 N

## REFERENCES

[1] Emmanuel Guilmineau , Oussama Chikhaoui, Ganbo Deng And Michel Visonneau (2011) -"Cross Wind Effects on a Simplified Car Model by a DES Approach"- Computer s & Fluids –pp.29-40.

[2] Joseph Katz (2008) –"Aerodynamics of Race Cars"- Fluid Mechanics– vol 63-pp.27-63.

[3] Simon Watkins And Gioacchino Vino (2007) – "The Effect of Vehicle Spacing on The Aerodynamics of A Representative Car Shape"- Wind Engineering and Industrial Aerodynamics -pp.1232-1239.

[4] Xin Zhang, Willem Toet And Jonathan Zerihan (2006)- "Ground Effect Aerodynamics of Race Cars" Applied Mechanics – Vol 95-pp.33-49.

[5] W. Kieffer, S. Moujaes and N. Armbya (2005)- "CFD Study of Section Characteristics of Formula Mazda Race Car Wings" Mathematical and Computer Modelling- Vol 43 – pp.1275-1287.