Analysis of Aerodynamic Characteristics In Front Wing of F1 Car Using CFD

Shubham Borole[1], Omkar Bhasale[2], Shubham Patil[3], Akshay Khot[4], Prof. Vivekanand Navadagi[5]
[1, 2, 3, 4, 5] Department of Mechanical Engineering
[1, 2, 3, 4, 5] Dhole Patil College of Engineering, Pune

Abstract- *In our Project we are improvising on the design of the front wing of a Formula 1 Vehicle. This helps in improving drag and down force which adds stability to the vehicle. Especially on turns/corners at high speeds. Having done CFD on the front wing we understand better designs as we learn the flow patterns and its effect our vehicle. Thus comparing different designs helped us to achieve higher efficiency of front wing.*

Keywords- Formula1, Front Wing, Drag, Lift, CFD, Stability

I. INTRODUCTION

- **INTRODUCTION TO CFD**

Computational fluid dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation. The technique is very powerful and spans a wide range of industrial and non-industrial application areas. Some examples are:

- Aerodynamics of aircraft and vehicles: lift and drag.
- Hydrodynamics of ships.
- Power plant: combustion in internal combustion engines and gas turbines.
- Turbo machinery: flows inside rotating passages, diffusers etc.
- Electrical and electronic engineering: cooling of equipment.
- Chemical process engineering: mixing and separation, polymer molding.
- External and internal environment of buildings.
- Marine engineering: loads on off-shore structures.
- Environmental engineering: distribution of pollutants and effluents.
- Hydrology and oceanography: flows in rivers and oceans.
- Meteorology: weather prediction.

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 engines, combustion chambers of gas turbines and furnaces. Furthermore, motor vehicle manufacturers now routinely predict drag forces, 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 developments in the CFD field is to provide a capability comparable with other CAE (computer-aided engineering) tools such as stress analysis codes. The main reason why CFD has lagged behind is the tremendous complexity of the underlying behaviour, which precludes a description of fluid flows that is at the same time economical and sufficiently complete. The availability of affordable high-performance computing hardware and the introduction of user-friendly interfaces have led to a recent upsurge of interest, and CFD has entered into the wider industrial community since the 1990s.

The development of Wind Tunnel technology, important though it has been, pales into insignificance alongside the rapid growth of Computational Fluid Dynamics (CFD). With a wind tunnel, experiments are made by blowing wind over a real object in a controlled environment and measuring the aerodynamic forces that arise. In CFD, the same experiment may be conducted in the form of a computer simulation. Although the equations that govern these computations have been understood since the 1930s, they are complex to solve and require the sort of computing power that has only become truly practical in the last 15 years.

- **INTRODUCTION TO F1**

For a sustained period of around 20 years, the teams in Formula 1 have ploughed money into the development of CFD, as it has been clear for a long time that mastery of this tool would be a prerequisite for success in the sport. Teams have sponsored the development of improved CFD techniques at top universities and they have also put money directly with the providers of commercial CFD codes to ensure that the considerable challenge of accurately simulating the aerodynamic behavior of a Formula 1 car, this has turned from an aspiration to a reality. It would be wrong to pretend that the development of subsonic CFD codes has been the sole responsibility of the Formula 1 industry, but no serious observer of the industry would deny that the combined investment of the teams has been very significant.

A major step forwards was the construction of the `Computational Aerodynamics Research Centre' in 2008. Prior to this, the wind tunnel and CFD aerodynamicists worked in the same building, which houses the wind tunnel. The construction of the centre enabled the team to increase its number of CFD aerodynamicists and also expand its computational power. To carry out the demanding CFD computations, the team owns an Xtreme-X2 supercomputer which has in excess of 4000 cores and 8Tbytes of memory. At its peak, it can provide a performance of 38TFlops. Next to all these hardware updates, the software used is frequently updated too. Not only are the commercial packages regularly updated to their latest versions, the in-house developed codes are constantly refined and improved.

Dr. Mori Mani, Senior Technical Fellow in CFD for Boeing explains: "With the reduced-cycle time CFD has the potential to become a primary contributor to vehicle design, as opposed to being primarily an analysis tool. In particular, the collaboration between Lotus F1 Team and Boeing has helped to accelerate further this type of numerical design, as well as improve the robustness of the tool suite through analysis of the challenging F1 geometry, which is typically more complicated than aircraft geometry."

A case study was carried out by Lotus Renault GP for Nissan to demonstrate the power of this newly developed optimization technique for road car design. Nissan provided the team with a virtual model of an existing Nissan road car as a test case and using this new CFD software they were able to optimize the external shape of the car and reduce drag by over 4%. This figure was then confirmed by Nissan using a different CFD code. This 4% drag reduction is a significant improvement and is achieved through geometry changes not easily identified using conventional CFD codes "The competitive nature of Formula 1 and the desire to extract greater levels of performance from the car has undoubtedly helped to develop the capabilities of CFD software at an ever-increasing rate," and applying CFD to simulate performance can be seen filtering down into passenger car development where more complete and detailed simulations are helping to improve safety at the same time as efficiency."

II. IDEOLOGY

Formula one is the one of event of motorsport currently referred to as the FIA Formula One World Championship In typical of journalist's language, F1 racing in the early 1980's, it's witnessed revolutionary changes because of the introduction of electronic driver aids, and active suspension for the sports cars. In the 1990s traction control and semi-automatic gearboxes were new additions to the sports car models. Aerodynamics has become the key to success in the Formula One sport and spends of millions of dollars on research and development in the field each year. The aerodynamic design has two primary concerns. Firstly, the creation of down force to help push the car's tires onto the track and improve the cornering forces. Secondly, to minimize the drag that caused by turbulence and act to slow the car down To create the down force to the car, the wings operate with air flow at different speeds over the two sides of the wing by having to travel different distances over its contour and form this creates a difference in pressure. This pressure can make the wing to move in the direction of the low pressure.

RESEARCH

- **Navier-Stokes Equations**

The Navier-Stokes equations (for an incompressible fluid) in adimensional form contain one parameter: the Reynolds number:

$$Re = \rho\, V_{ref} L_{ref} \,/\, \mu$$

It measures the relative importance of convection and diffusion mechanisms

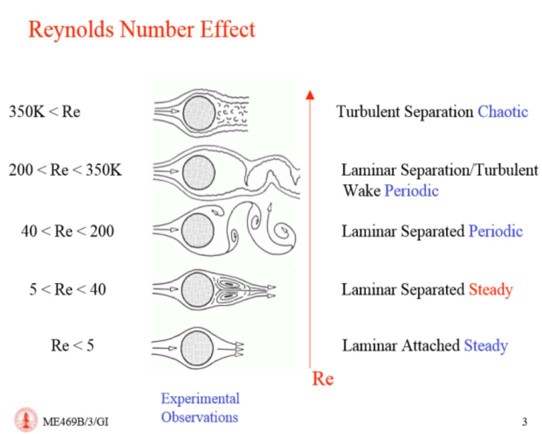


Fig: Reynolds Number Effect

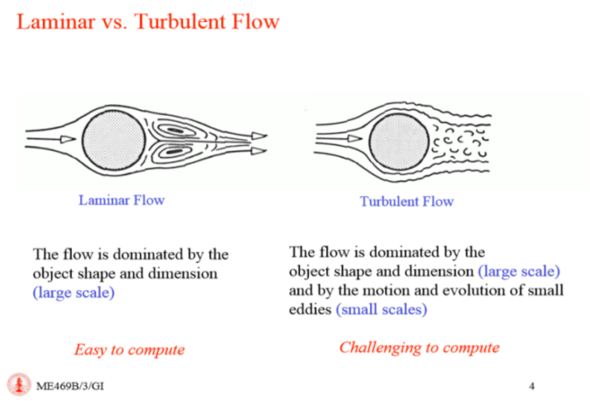


Fig: Laminar Vs Turbulent Flow

The Multi-Element Airfoil

Because of the restrictions due to the FIA F1 technical regulation, the down force needed has to be obtained with a limited surface. So, to get the right balance on the car, introducing a multi–element airfoil is required. The first element sets a minimum amount of down force and the second element is a movable part of which the angle of attack may be changed. In this way, teams are able to modify the balance of the car easily during testing sessions and grand prix.
The second element is used to being set at an angle of attack between 10 and 25 degrees. To avoid any vortex, the first element was designed as a rectangular platform and the modifications of the chord along the span were made smoothly for the second element. Indeed, if corners had occurred along the span of the second element to decrease the chord, vortices would have been created.

Turbulence Creation

Unsteady, irregular motion in which transported quantities (mass, momentum, scalar species) fluctuate in time and space. Identifiable swirling patterns characterize turbulent eddies. Enhanced mixing (matter, momentum, energy, etc.) results Fluid properties and velocity exhibit random variations. Statistical averaging results in accountable, turbulence related transport mechanisms. This characteristic allows for turbulence modelling. Contains a wide range of turbulent eddy sizes (scales spectrum). The size/velocity of large eddies is on the order of mean flow. Large eddies derive energy from the mean flow Energy is transferred from larger eddies to smaller eddies in the smallest eddies, turbulent energy is converted to internal energy by viscous dissipation.

Overview of Computational Approaches:

Reynolds-Averaged Navier-Stokes (RANS) models Solve ensemble-averaged (or time-averaged) Navier-Stokes equations. All turbulent length scales are modelled in RANS. The most widely used approach for calculating industrial flows. ‹Large Eddy Simulation (LES) solves the spatially averaged N-S equations. Large eddies are directly resolved, but eddies smaller than the mesh are modelled. Less expensive than DNS, but the amount of computational resources and efforts are still too large for most practical applications. Direct Numerical Simulation (DNS) theoretically, all turbulent flows can be simulated by numerically solving the full Navier-Stokes equations. Resolves the whole spectrum of scales. No modelling is required. But the cost is too prohibitive! Not practical for industrial flows -DNS is not available in Fluent

Standard k–ε (SKE) model

The most widely-used engineering turbulence model for industrial applications Robust and reasonably accurate Contains submodels for compressibility, buoyancy, combustion, etc. Limitations .The ε equation contains a term which cannot be calculated at the wall. Therefore, wall functions must be used. Generally performs poorly for flows with strong separation, large streamline curvature, and large pressure gradient.

The run time achieved by these models is purely based on the aerodynamic performance of the car. The section outlines the history of road vehicle aerodynamic design, as well as factors affecting the aerodynamic design of these cars including drag, physical flow characteristics and their impacts on racing and the use of ground effect. Finally, this section looks at the aerodynamic differences and focusing on Reynolds number and what differences this parameter can imply.

- **Pros & Cons of k-ε model**

Positives of k-ε model

- Simple
- Affordable
- reasonably accurate for wide variety of flows (without separation)
- History effects

Negatives of k-ε model

- Overly diffusive
- Cannot predict different flows with the same set of constants (universality).

- Source terms are stiff numerically.
- Not accurate in the region close to no-slip walls where k and ε exhibit large peaks (DNS and experimental observations).

Forces acting on the vehicle.

- **Lift**

Lift is the components of the pressure and wall shear force in the direction normal to the flow tend to move the body in that direction. It can prevent the object from flying to the air when we use the negative lift coefficient. The pressure difference between the top and bottom surface of the wing generate an upward force that tends the wing to lift. For the slender bodies such as wings, the shear force acts nearly parallel to the flow direction, thus its contribution to the lift is small. The lift force depend on the density ( ), of the fluid, the upstream velocity (V), the size, shape, and orientation of the body, among other things, and it is not practical to list these force for a variety of situations. Instead, it is found convenient to work with appropriate dimension less numbers that present the drag and lift characteristics of the body.

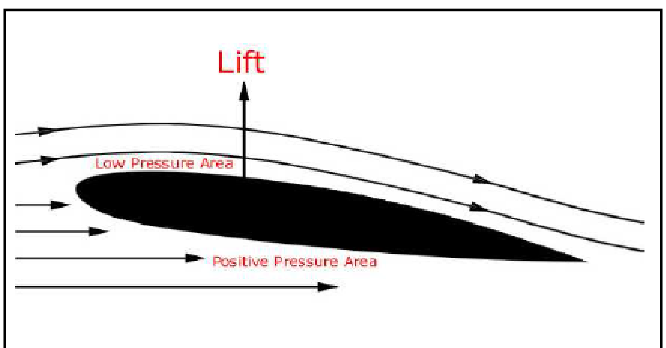


Figure 2.1: Lift direction

Fig: Lift Direction

Where A is ordinarily the frontal area (the area projected on a plane normal to the direction of flow) of the body ½ V2is the dynamic pressure and CL is lift force.

$$F_L = \frac{C_L}{2\rho V^2 A}$$

- **Drag**

Drag is the aerodynamic force that is opposite to the velocity of an object moving through air or any other fluid. Its size is proportional to the speed differential between air and the solid object. Drag comes in various forms, one of them being friction drag which is the result of the friction of the solid molecules against air molecules in their boundary layer. Friction and its drag depend on the fluid and the solid properties. A smooth surface of the solid for example produces less skin friction compared to a rough one. For the fluid, the friction varies along with its viscosity and the relative magnitude of the viscous forces to the motion of the flow, expressed as the Reynolds number. Along the solid surface, a boundary layer of low energy flow is generated and the magnitude of the skin friction depends on conditions in the boundary layer.

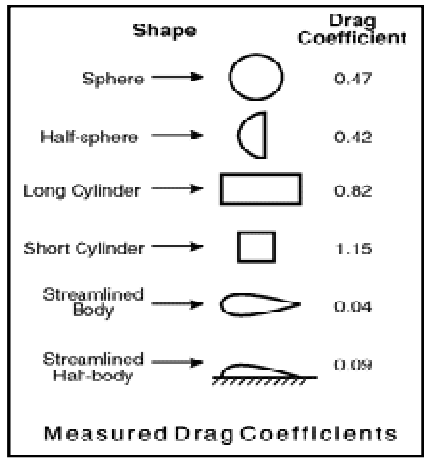


Fig: Measured Drag Coefficients

Additionally, drag is a form of resistance from the air against the solid moving object. This form of drag is dependent on the particular shape of a wing. Due to the change of direction of air around the wing, a vortex is created where the airflow meets unchanged, straight flow. The size of the vortex, its drag strength increases with an increasing angle of attack of the aerofoil. The amount of drag that a certain object generates in airflow is quantified in a drag coefficient. This coefficient expresses the ratio of the drag force to the force produced by the dynamic pressure times the area. Therefore, a Cd of 1 denotes that all air flowing onto the object will be stopped, while a theoretical 0 is a perfectly clean air stream. At relatively high speeds of high Reynolds number (Re> 1000), the aerodynamic drag force can be calculated by this formula:

$$\boxed{F_d = \frac{C_d}{2\rho V^2 A}}$$

Where
Fd = Force of drag
 = Density of the air
V2= Speed of the object relative to the fluid (m/s)
A = Reference surface area

Cd = Drag of coefficient

Other sources of drag include wave drag and ram drag. For the wave drag, it is unimportant for normal race cars as it occurs when the moving object speeds up to the speed of sound. Ram drag on the other hand is the result of slowing down the free airstream, as in an air inlet.

- **Down Force**

Aerofoil in motorsports are called wings where can generate high down force by having a high angle of attack, thus increasing the drag of the aerofoil. An aerofoil's operation can be easily explained when consider a wing in a steady laminar flow of air. As down force generating aerofoils are mostly designed with more thickness on the lower side, the lower airstream is slightly reduced in surface, hence increasing the flow speed and decreasing the pressure. On top of the wing, the airspeed is lower, and thus the pressure difference will generate a downward force on the wing.

$$F = m_a = m\frac{(V_1 - V_0)}{(t_1 - t_0)} = M\frac{dV}{dt}$$

This shows that a force, causes a change in velocity, V, or also a change in velocity generates a force. Note that a velocity is a vectorial unit, having a speed and a direction component. So, to change of either of these components, force must be imposing. And if either the speed or the direction of a flow is changed, a force is generated.

Down force is often explained by the "equal transit time" or "longer path" theory, stating that particles that split ahead of the aerofoil will join together behind it. In reality however, the air on the longer side of the wing will flow much faster, further increasing the down force effect Because of the complexity, today's formula one cars are designed with CFD (computational fluid dynamics) and CAD (computer aided design) that allows engineers to design a car, and immediately simulate the airflow around it, incorporating environmental parameters like traction, wind speed and direction, and much more. From commercial CFD software, the drag coefficient, Cd and lift coefficient, CL can be known to create downforce for the formula one car.

The Concept and Usage of CFD

The development of modern computational fluid dynamics began with the advent of the digital computer in early 1950s. It uses finite difference methods and finite element method as the basic tools used in the solution of partial differential equations in general and computational fluid dynamics. The fundamental basic of almost all computational fluid dynamics problems are the Navier-Stokes equations, which define any single-phase fluid flow. These equations can be simplified by removing terms describing viscosity to yield the Euler Equation and removing terms describing vortices.

- **Use of software's**

- **CAD**

CATIA is one of the leading CAD programs to date. It is a solid modelling program which has good interconnectivity with many grid generation systems, and avoids the interface problems outlined by Kellar et al. (1999) regarding the methods previously used by CAD programs. It has an easy to use system of part design, followed by integrating parts together into an assembly.

- **Mesh Generation**

Mesh generation is a very important part of the CFD process. The fineness or coarseness of a grid determines the accuracy of solution as well as rate of convergence. The finer the grid, the more accurate the result, however the computational time increases significantly with a finer grid. There is a variety of meshing methods available including Cartesian, structured, unstructured, and hybrid grids. These different types of grids give different results. For example to calculate accurate boundary layer flow, structured grids are the most useful with rectangular cells, which can be elongated to fit the profile of the shape, but compressed in a normal direction in order to maintain grid fineness to obtain accurate results for boundary layer flow. The weakness of structured grids, however, is that rectangular cells do not easily fit well around complex shapes. To work around both of these problems, a hybrid grid may be used. Hybrid grids of tetrahedral elements used in combination with rectangular elements near surfaces provide a significant reduction in meshing time, and are flexible in dealing with complex geometries.

- **Flow Solving**

The CFD solver used was Fluent (ANSYS 15), which has easy interface with GAMBIT and gives fairly accurate results for low speed incompressible flow when used correctly. The problem with CFD is that it can be a great benefit in understanding flow motion; however it can represent a real lack of information if used inadequately, giving appealing pictures but wrong answers. Using CFD requires knowledge and experience of its user. If not, it will lead to incorrect results and poor accuracy in flow visualization (Martinez 2004). This is why setting up the solution is so important. Boundary conditions, as well as flow conditions and properties must be established correctly in order to obtain the best results possible.

- **Verification and Validation**

Verification and validation of CFD has been the biggest deterrent from its widespread use until recent times. Verification of a CFD result involves a process to ensure that the CFD result is the most accurate that can be generated by the solver (Versteeg et al. 2007). Verification allows the user to quantify different types of error, and minimize where possible. There is a process to verifying the use of CFD results that Martinez (2004) describes as necessary for a proper analysis. The steps to this process are as follows; examine iterative convergence, examine consistency, examine grid convergence and examine temporal convergence. The error of using unconverged results can occur because of an ineffective grid, or not setting a low enough residual minimum. To avoid this, a physical solution parameter can be observed as the solver works through iterations. Once that parameter remains unchanged over many iterations (as well as the residuals reaching an acceptable low), the solution can be taken as converged. As well as making sure the solution is correctly converged; uncertainty due to the discretization process needs to be analysed. This uncertainty represents the difference between the discretized equations solved and the true partial differential equations, this is called discretization error. The idea of validating of CFD results makes sure that the CFD solver is solving the correct environment characteristics and flow properties, to make sure that the CFD simulation is a true representation of the real

DESCRIPTION OF THE FLOW AROUND THE FRONT WING

- **FLOW OVER THE WING SPAN**

The first element of the wing is a rectangular element. The flow around it is a classical flow around an inverted airfoil which benefits from ground effect. The latter is emphasized at the centre of the span as the regulations allow the team to build body device beneath 0.1 m above the reference plane (so between 0 and 0.1m) in an area that does not exceed 0.25 m from the centre line of the car.

The second element chord is usually reduced at the centre of the span to decrease the deflected flow downstream and to avoid interaction between the nose and the second element. Finally a Gurney flap may be fixed to the aft flap trailing edge to increase the down force in an ease and quick way. The Gurney flap causes a lower pressure area just behind itself which sucks the lower flow closer to the wing surface The Gurney causes some extra drag as well, but the wing can be run at a higher angle of attack and so can produce more downforce.

- **FLOW AT THE TIPS**

The main aim of the tips is to deflect the flow from the front wheels. The latter influence the air flow in a way that the air close to the tyre surfaces may be a reverse flow compared to the overall flow coming from upstream. There would be surface viscous effects too on the sides of the tires but this may not be included in this study as this effects should influence the downstream flow rather that the front wing environment.

Then, if the wing tip has an upper flat plate, a high pressure area can be created on it, thanks to the viscous effect due to the rotating wheel if the horizontal surface carrying the load is sufficiently close to the tyre surface. Some horizontal surfaces may be designed as close as possible to the tyre surface (taking account of the fact the wheels have to rotate to permit the car turning) to create high pressure on their upper surface.

- **Flow over Inclined Flat Plates**

As the vortex becomes flat when it gets close to the lower surface of the inclined flat plate, it should be tested by a higher angle of attack. However, the vortex might get slower because of the friction next to the lower surface of the flat plate.

Then, as the vortex flow is disrupted by the flow from the rotating wheel and that the latter induces high pressure area at both upper and lower surface of the flat plate, the downforce created by the flat plate would be risen if it was set closer to the ground as the wheel is further at this location. A way to improve this device would be to higher the vortex size. To perform this improvement, the flat plate leading edge should extent in width.

This would imply to cut the vertical airfoil just upon the upper surface of the first element of the two-element wing. However, this will not alleviate the down force provided by the multi-element airfoil too much as it would only affect the super pressure upon it. The design would have to be smooth as the aim is to prevent the formation of vortices but the one from the flat plate.

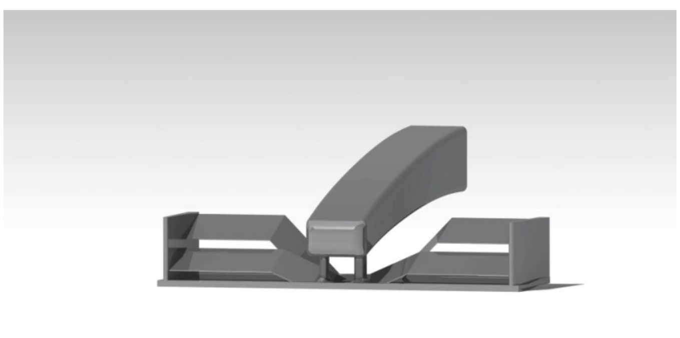
- **VERTICAL AIRFOILS**

In this case, the aim of this small airfoil would be to dilute the super pressure located on the wheels and maybe to decrease drag but it would not be significant. This would in fact transfer some drag from the tyre to the front wing. To allow more air to be deflected inboard the wheels, the vertical airfoil should be twisted at its bottom near its trailing edge to increase the gap between the tyre and this airfoil. A greater amount of air may be deflected in this gap so that less air would have to hit the tyre and so again in drag may be obtained.

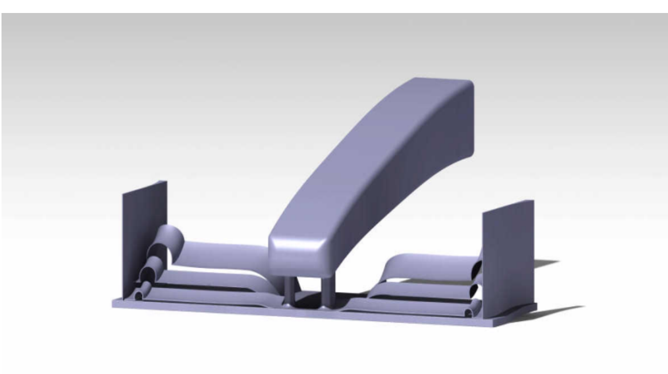
To avoid the wing tip vortex at the top of the vertical airfoil, a winglet may be added at the top of the vertical airfoil. It should be done inboard (at 0.3 meter above the reference plane, i.e. at the limit allowed by the regulation or by following the airfoils' elements upper surface shape to drive the air) and outboard.

It could be possible to control the vortex at the tips (so at the top and at the bottom) by playing with the tip curvature of the vertical airfoil.

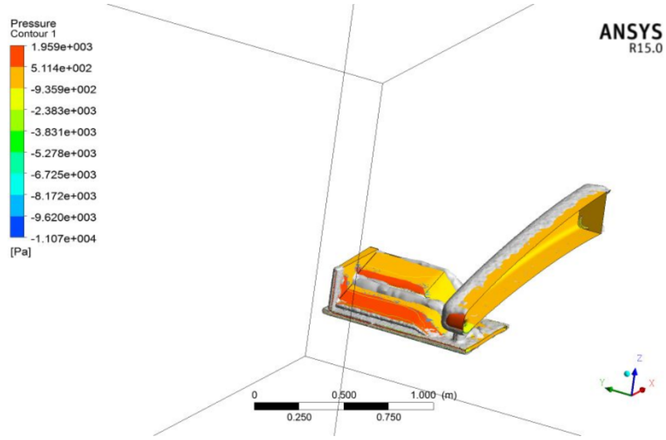
IV. DESIGN&ANALYSIS

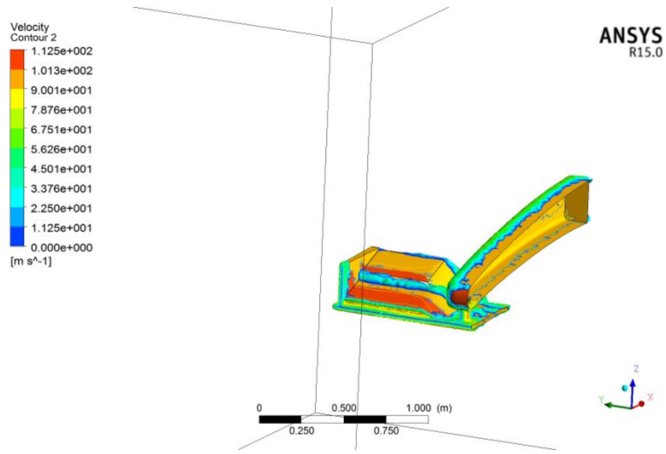


Primary design

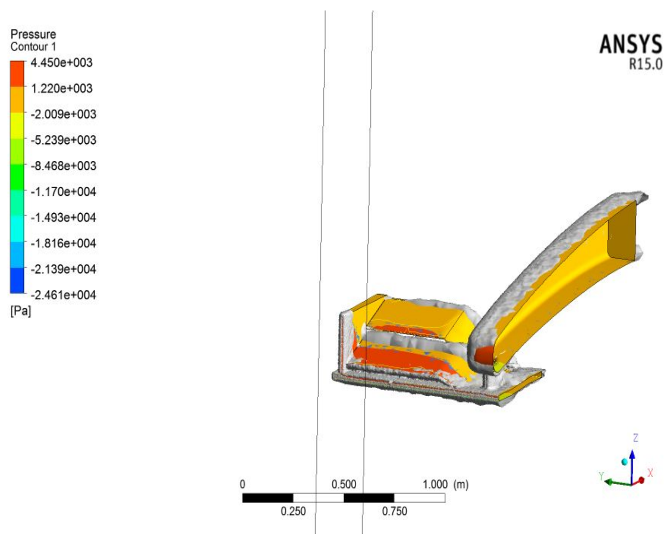


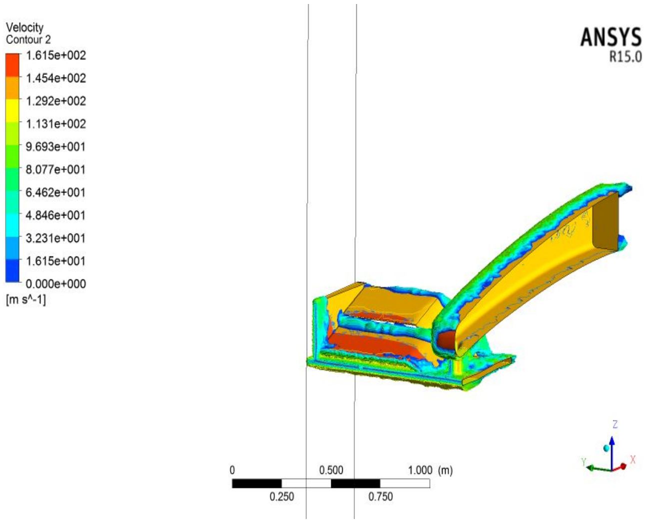
Improved design



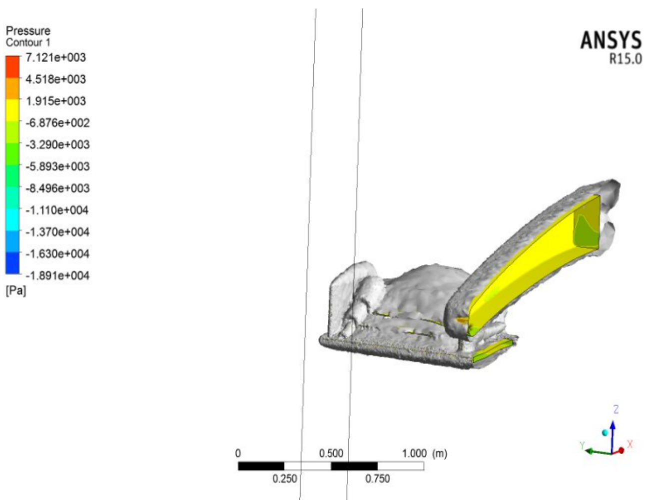


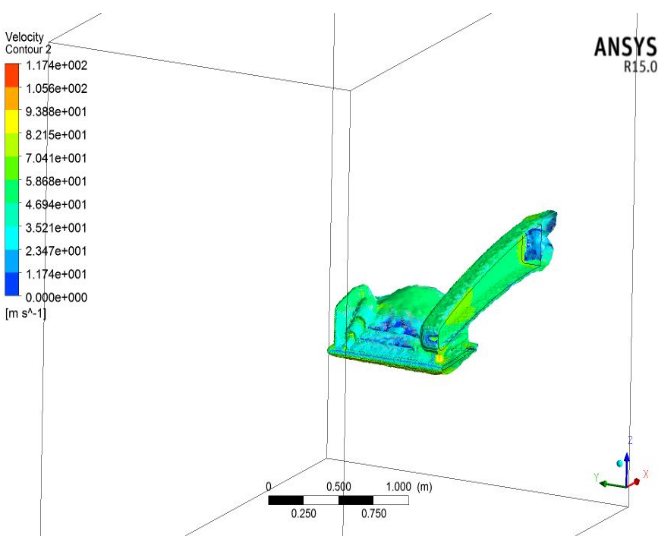
Pressure vortices& Velocity vortices at 200 km/hr



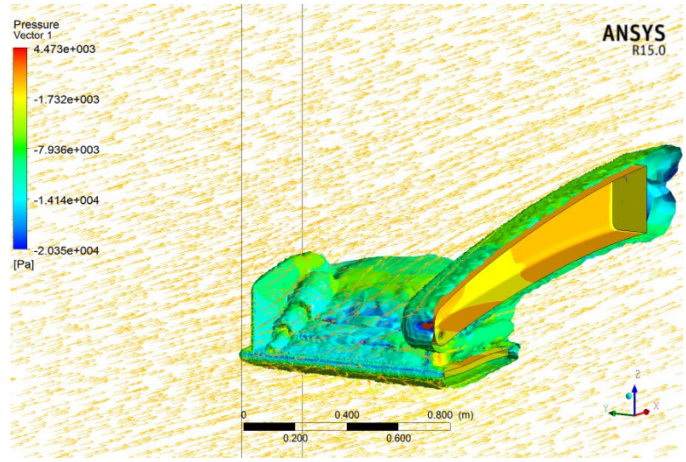


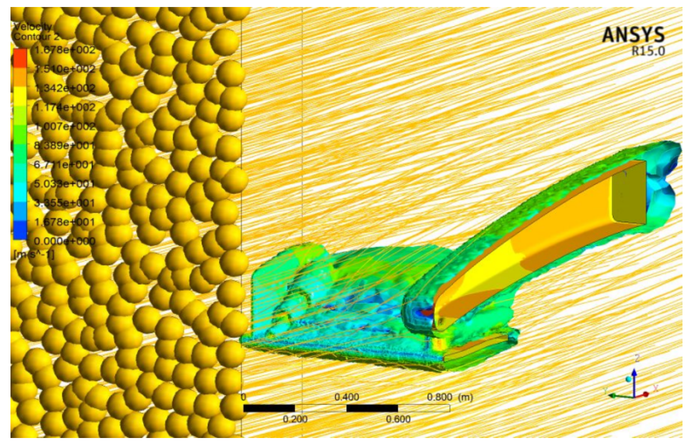
Pressure vortices& Velocity vortices at 300 km/hr





Pressure vortices& Velocity vortices at 200 km/hr





Pressure vortices& Velocity vortices at 300 km/hr

Comparison table @ 200KMPH

Sr. no	Parameters	Primary design	Improved design
1)	Coefficient of lift	-0.7300	-0.8478
2)	Coefficient of drag	0.5624	0.4288

When diagnosed it can be seen that the lift increases in the negative direction thereby there is increase in the downforce on the vehicle. The improved design has even better drag results as compared to the primary design.

Comparison table @ 300KMPH

Sr no	Parameters	Existing design	Improved design
1)	Coefficient of lift	-0.7278	-0.8953
2)	Coefficient of drag	0.51072	0.4056

At the speed of 300 kmph.the down force acting on the vehicle further increases thereby improving the efficiency of the design even further.

VI. CONCLUSION

The aerodynamic of racing cars can be improved by using computational fluid dynamic (CFD) tools. These tools provide results quite accurate within very short time scales. Results of simulations are generally not grid independent 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 and track testing. Without validation, results of simulations are indeed of no value.

The set of results and observations included in this report suggests that CFD can be a very useful tool to support the aerodynamic design of race cars. The most important requirement for the production of accurate results is the mesh resolution, with a minimum advised for any calculations looking to determine the aerodynamic forces produced by the vehicle increasing with the complexity of the car aerodynamic. The choice of the appropriate turbulence model needs further investigation, as the different available formulations were not applied to the large meshes. However, the predicted flow patterns suggest that the specific k-ε models perform better when predicting the separation point on the aerodynamic surfaces and when dealing with transition and generates the following advantages:

1) Increases tires capability to produce cornering force.
2) Stabilizes vehicles at high speed.
3) Improves braking performance.
4) Gives better traction

Having more negative force than having less drag can be more important for race cars since driving safely is always number one priority. Separation effects in general than the basic k -ε models. The addition of a transition model to these calculations could greatly improve the results.

V. FUTURE WORK

- To perform parametric studies of various ground clearances of front wing to see its effects on the aforementioned coefficients.
- The consideration of thermal temperature gradients around the race car and its effects through the air density on these coefficients should be studied.
- A deeper research in optimization algorithms should be done, especially to understand how to modify the parameters of the optimizer to fit better the aim of the engineer.
- With more computing resources and more accurate mesh, another different perspective should be developed, tested and correlated, to maximize the efficiency of the loop and to confirm that the success of this experiment is not only restricted to front wing developments.

REFERENCES

[1] International Journal of Application or Innovation in Engineering &Management (IJAIEM). STUDY OF FRONT BODY OF FORMULA 1 CAR FOR AERODYNAMICS USING CFD
Web Site: www.ijaiem.org Email: editor@ijaiem.org, Volume 3, Issue 3, March 2014 ISSN 2319 – 4847.

[2] A. Muthuvel, N. Prakash, J. Godwin John International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 NUMERICAL SIMULATION OF DRAG REDUCTION IN FORMULA ONE CARS
National Conference on Advances in Engineering and Technology (AET- 29th March 2014)

[3] Aniruddha Patil1, Siddharth Kshirsagar1 and Tejas Parge1
ISSN 2278 – 0149STUDY OF FRONT WING OF FORMULA ONE CAR USING COMPUTATIONAL FLUID DYNAMICS www.ijmerr.com Vol. 3, No. 4, October 2014 © 2014 IJMERR. All Rights Reserved.

[4] Dr. Ugur GuvenProfessor of Aerospace Engineering (PhD)
Nuclear Science and Technology Engineer (M.Sc.)
Flow Analysis & Simulation of Formula SAE Vehicle using CFD Techniques

[5] An Introduction to Computational Fluid Dynamics
THE FINITE VOLUME METHOD
Second Edition
H K Versteeg and W Malalasekera

[6] CRANFIELD TEAM F1: THE FRONT WING
Supervisor: P A RUBINI
September 2003