CFD ANALYSIS OF FRONT WINGS IN F-1 CARS

A Nishanth Kumar[1],Sudhakar Uppalapati[2],K. Veera Raghavulu[3],Sandip Bhattacharya[4],Sanika Anuradha[5]

[1]Assistant Professor, Mechanical Department , Marri Laxman Reddy Institute of Technology & Management, Dundigal, Hyderabad,500043, India

[2]Associate Professor, Mechanical Department, Marri Laxman Reddy Institute of Technology & Management, Dundigal Hyderabad,500043, India

[3]Assistant Professor, Mechanical Department , Marri Laxman Reddy Institute of Technology & Management, Dundigal, Hyderabad,500043, India

[4]UG Student, Mechanical Department , Marri Laxman Reddy Institute of Technology & Management, Dundigal, Hyderabad,500043, India

[5]UG Student, Mechanical Department , Marri Laxman Reddy Institute of Technology & Management, Dundigal, Hyderabad,500043, India

nishanth.aluri@gmail.com[1], sudhakaruppalapati@mlritm.ac.in[2], raghava.gnitc@gmail.com[3] rajsan250@gmail.com[4] , sanikaanuradha248@gmail.com[5]

Abstract

In this project, flow analysis of two F-1 front wing models, McLaren and Red bull models, has been done. CFD analysis has been done, and the drag force was determined in the solver. Then the drag coefficient has been obtained for both models and compared with experimental results previously achieved in a random design. The distribution of pressure, velocity, and turbulence kinetic energy has been achieved, and finally, by comparing the results, the need for a better design has been shown significant

Keywords: *Front wing, CFD, Formula-1, Drag force, Drag coefficient*

1 Introduction:

Formula One, abbreviated as F-1, an international sports car racing event, where we see single-seat, open-wheel, open-cockpit professional motor racing contests. F-1 racing is regulated and authorized by a single world-wide governing body known as the FIA - Fédération Internationale de l' Automobile. The terminology of this 'Formula' is derived with regards to a set of rules that cars and drivers are taking part in the event must follow.

The overall purpose of this contest (F-1) is to appoint the winner of the race as victorious. The participant who reaches the finish line first after completing several laps, that the management has already fixed, is considered as the winner. F-1 racing came into existence during the 1920-30s in Europe.

ISSN: 2005-4238 IJAST

Figure 1. Formula-1 Car

Aerodynamics plays an essential factor in criterion in the complete arrangement of an F-1 car. It consists of an air duct panel amongst the front-wheels and side-panels. Considering an example, extra speeds of more than 2 or 3 horse-powers can be added. A team invests approximately 20% of its overall financial budget towards the proper understanding of the aerodynamics of the car. Modern F-1 cars can push the limits to the corners, a lot faster compared to the normal, daily-used auto, and this cannot be possible without considering the down-force. The majority of the precision work is done by performing calculations and mostly rely on the wind tunnel tests to get accurate results of the wings under study and the wind-deflectors till the last millimeter. It also enables short-braking distances and large cornering speeds. Down-force produced eighty percentile of the necessary grip required by the car. Without sliding off the track, F-1 cars can withstand centrifugal forces up to the range of 4G, mainly because of aerodynamic designs that allow high cornering speeds. Front wings work in a different procedure when compared to an airplane wing. The aircraft wing generates upward-lift, whereas the front side generates down-force to prevent a car from taking off. In other words, air-foil on front-wings is profiled so that the vehicle remains on the ground and assures the wheels stay in contact with the ground surface. Since front-wings are the first part of the car which comes in contact with air, the front wings determine F1 car's overall aerodynamics. The other property is to direct the airflow in the required order, best suited for the car's aerodynamics.

CFD (or Computational Fluid Dynamics), one of the extensions of fluid mechanics, involves numerical finite element analysis of models and specific structures to understand the processes of fluid flow visually and hence solve related problems. Computer software is used for calculations that are necessary to work out visually the fluid's free flow and the behaviour of fluids with a surface that is subjected to certain boundary conditions. High-speed supercomputers provide accurate solutions that are achieved and are necessary to simulate the largest and highly complex and challenging problems.

CFD is applied to various research problems and techniques like multiple fields of study in this industry, that includes aerodynamics and analyzation of aerospace problems, the study of the weather conditions, also natural sciences and studies like environmental-engineering, design of systems in industries and their analysis, the study of fluids in biological engineering, areas of study in the transfer of heat mainly in the engine and the combustion analyzer, etc.

1.1 Literature review

The main focus of this study is to determine the drag coefficient of McLaren and new design Red bull 2020 models of the front wing and compared with experimental values obtained from the references to check for better performance and also check the physical behaviour of the wings in the given input conditions.

(1) Studies on auto racing that began in the late 1990s. Many wing profiles are there, but not all are suitable for racing cars. Aerodynamic front wings are very significant. These results were obtained from more than a dozen aerodynamic software but accurate results obtained from CFX. Simulation of viscous flow of Katz et al, shows the flow around a generic Indy car racing. The multi-element front wing showed different applications of the wing element.

(2) On the front wing, endplates race car affect the flow characteristics of the wing. Overall reported almost the same characteristics with Mokhtar. Endplates weaken the wingtip vortices and wakeless disability compared with wings without endplates. Prices show that the endplates are also essential to control the direction of flow. In his Formula SAE racing car, he reported a negative effect on the tires

without wingtips, which he said could be avoided by guiding air around the tires. He stated that removing the rear wing endplates causing loss of lift but no remnants of the stall characteristics.

2 2. Materials and Methods

2.1 Software's used

1) Solid works (for designing)

2) Ansys CFX (for CFD analysis)

2.2 Methodology

First, the designs have been made in Solid works by following the standard dimensions specified for F-1 cars. The plans were imported in Ansys workbench, where we first gave the domain around each model was created. Then we went for meshing of the models wherein the mesh parameters we set the element size as 0.2, Quads as mesh type, and fine mesh as the mesh quality. After this, in the solver, the boundary conditions were specified at the inlet and outlet.

2.3 Models

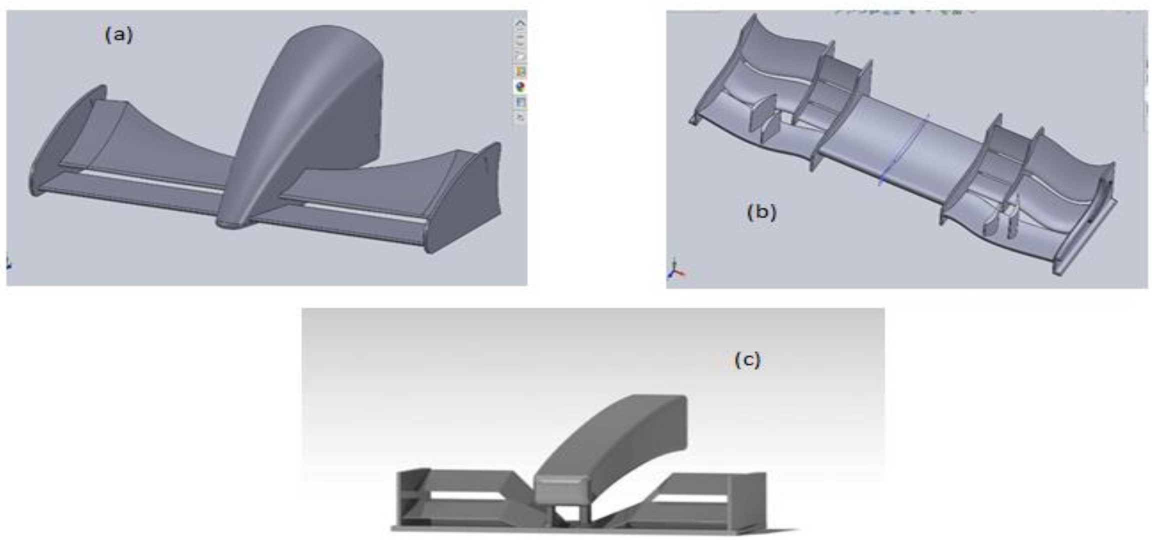

Fig : 2 (a),(b),(c) shows the 3-D models of McLaren, Redbull and experimental models respectively

2.4 Table for boundary conditions

Sl. No.	Conditions	McLaren	Redbull	Experimental
1	**Inlet** **(Speed in kmph)**	i) 200 ii) 300	i) 200 ii) 300	i) 200 ii) 300
2	**Outlet**	0 Pa static pressure	0 Pa static pressure	0 Pa static pressure
3	**Boundary**	Free slip condition	Free slip condition	Free slip condition

3 Results and Discussion

3.1 Plots

Y-axis indicates the magnitude of forces acting on the wing in Newton, but green colour lines in the plot denote the main drag force.

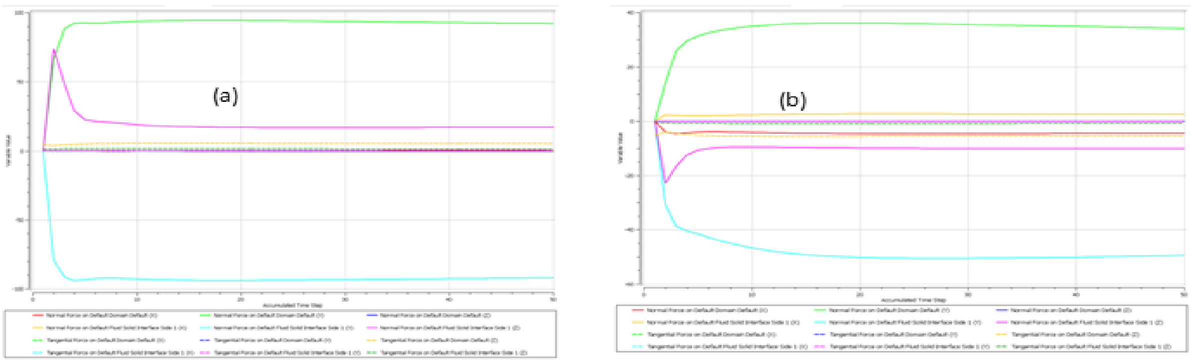

Fig 3. (a) and (b) shows max value of drag force obtained by McLaren at 200km/hr and 300km/hr respectively.

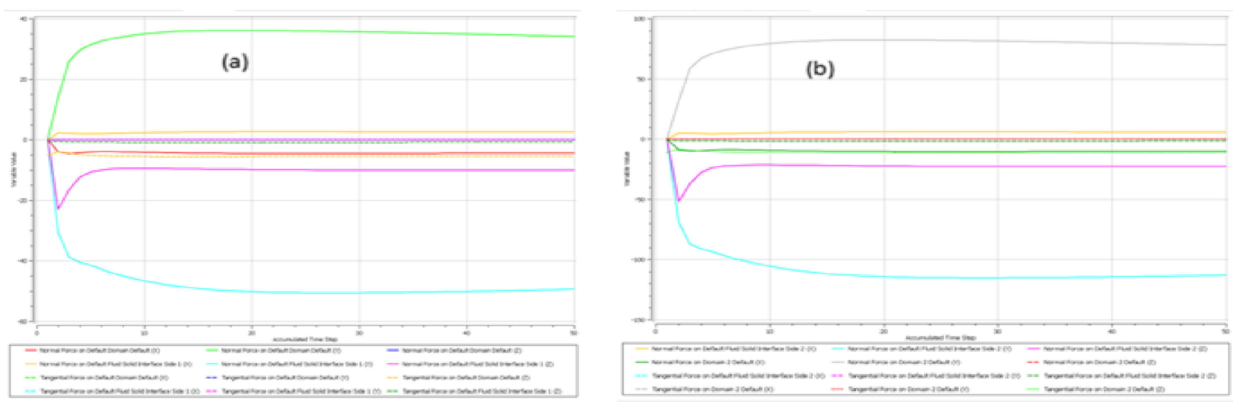

Fig 4. (a) and (b) shows max value of drag force obtained by McLaren at 200km/hr and 300km/hr respectively.

3.2 Pressure contours

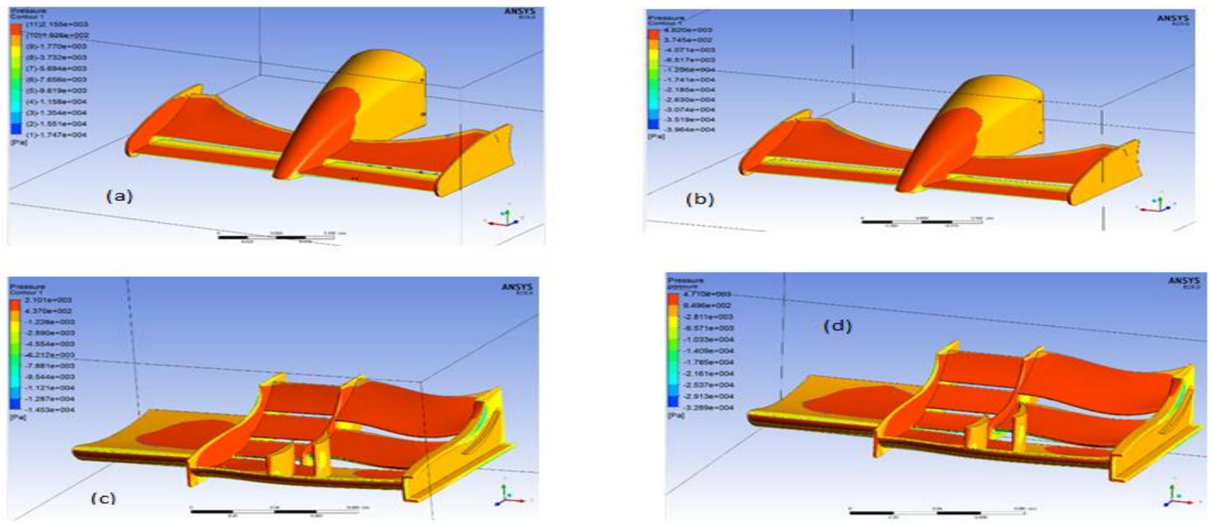

Figure 5. (a), (b) shows Pressure contours for McLaren (at 200 and 300 km/hr respectively) and (c), (d) shows the same parameter for Redbull (at 200 and 300 km/hr respectively)

When air flowing through an object pushes harder against the front than the back, backward force is created. Due to this separation, the turbulent air behind the airfoil decreases in pressure, which leads to an increase in pressure drag. For McLaren at 200 the max pressure obtained is $2.15*10^3$ Pa and at 300 the value is $4.82*10^3$ Pa, and for Redbull at 200 the max pressure value is $2.1*10^3$Pa and at 300 the value is $4.71*10^3$ Pa.

3.3 Turbulence Kinetic Energy contours

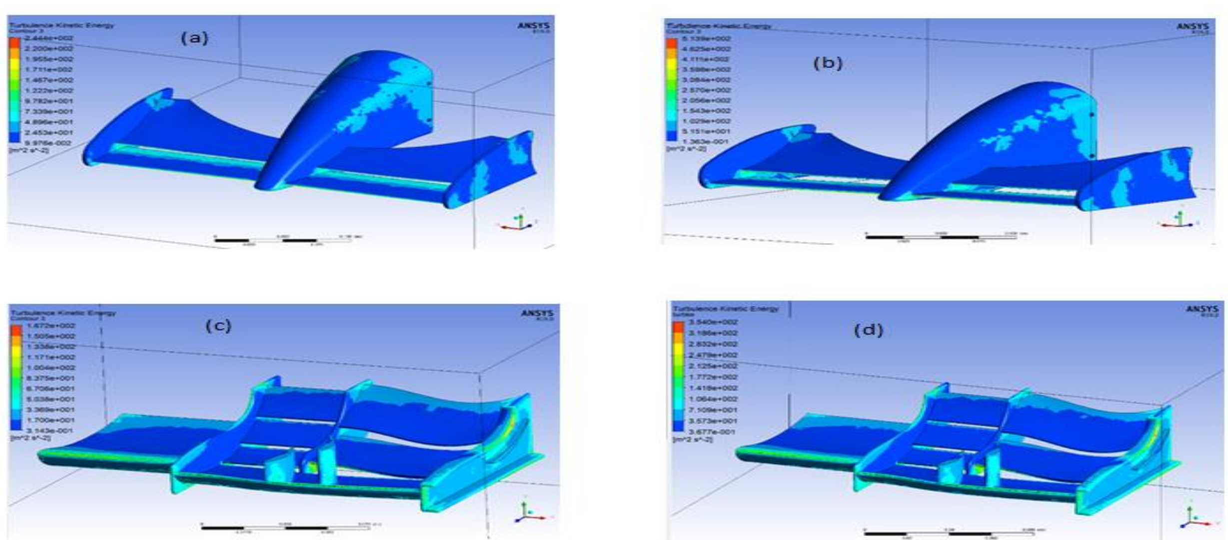

Figure 6. (a), (b) shows Turbulent Kinetic Energy contours for McLaren (at 200 and 300 km/hr respectively) and (c), (d) shows the same parameters for Redbull (at 200 and 300 km/hr respectively)

Turbulence occurs due to excessive kinetic energy in parts while in a fluid flow. In turbulent flow, unsteady vortices of many sizes are formed which interact with one other, Thus drag due to this type of friction increases. For McLaren at 200 the max turbulence obtained is $2.44*10^2$ m^2/s^2 and at 300 the value is $5.13*10^2$ m^2/s^2, and for Redbull at 200 the max turbulence value is $1.67*10^2$ m^2/s^2 and at 300 the value is $3.54*10^2$ m^2/s^2.

3.4 Velocity contours

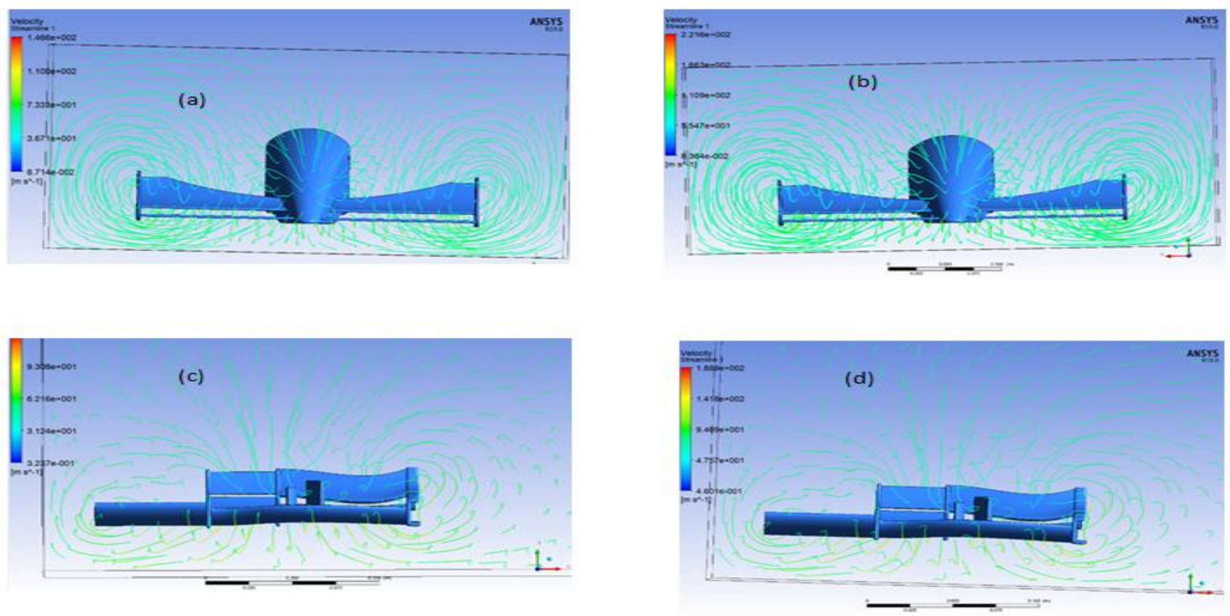

Figure 7. (a), (b) shows Velocity contours for McLaren (at 200 and 300 km/hr respectively) and (c), (d) shows the same parameters for Redbull (at 200 and 300 km/hr respectively)

When object collides with air particles, drag comes into play. Thus when the object travels faster, it comes into contact with more air particles and hence increases the drag. Thus an increase in velocity will lead to an increase in the drag forces. For McLaren at 200 the max velocity obtained is $1.46*10^2$ m/s and at 300 the value is $2.21*10^2$ m/s, and for Redbull at 200 the max velocity value is $0.93*10^2$ m/s and at 300 the value is $1.88*10^2$ m/s.

3.5 Results

1) For McLaren at speed 200 km/hr, the drag-force obtained was 18.3083 N.

2) For McLaren at speed 300 km/hr, the drag-force obtained was 40.8026 N.

3) For Red bull at speed 200 km/hr, the drag-force obtained was 10.7279 N.

4) For Red bull at speed 200 km/hr, the drag-force obtained was 24.2672 N.

5) The formula for finding the Drag coefficient in each model is :

$C_D = (2F) / (\rho A V^2)$

Where,

C_D = drag-coefficient

F = drag-force

ρ = Air density

A = Area of body subjected

V = Velocity of body

6) The drag coefficients for McLaren model at 200 km/hr and 300 km/hr were found to be 0.4917 and 0.4868 respectively.

7) The drag coefficients for Red bull model at 200 km/hr and 300 km/hr were found to be 0.2987 and 0.2712 respectively.

At velocity : 200 km/hr

Sl.No.	Parameters	Exp. Results	McLaren	Redbull
1.	Coefficient of Drag	0.5624	0.4917	0.2987

At velocity : 300 km/hr

Sl.No.	Parameters	Exp. Results	McLaren	Redbull
1.	Coefficient of Drag	0.5107	0.4868	0.2712

4. Conclusion

From the results Red bull front wing model experiences the least drag forces. The designs of each model have been done in Solid works by taking into consideration the design parameters. We obtained the drag forces of each model from Ansys CFX solver that is used for solving CFD problems efficiently. From the obtained drag forces, we calculated the drag coefficient using the relation between drag force and coefficient of drag

References

1. A. Muthuvel, N. Prakash, J. Godwin John, Title : “Numerical simulation of the reduction of barriers in a Formula One car of the National Conference on Advances in Engineering and Technology”.
2. Aniruddha Patil1, Siddharth Kshirsagar1 and Tejas Pargel, Title : “Study of a formula one car's front wing using computational fluid dynamics”.
3. Dr.Ugur Guven, Title : “Flow Analysis & Simulation Formula SAE vehicle using CFD techniques”.
4. Title : “An Introduction to Computational Fluid Dynamics”, by H K Versteeg and W Malalasekera.
5. Title : “Analysis In Front Wing Aerodynamic Characteristics of F1 Car Using CFD”, by Shubham Borole.