CFD Study of Effect of Yaw characteristics in Wing-and-Nose Section of a Formula-1 Car and Bluff body Diffuser Analysis

Shreyas Ravi

(18029106)

Abstract

Influence of yaw in a racing car is high and is discussed in this paper. Yawing condition is experienced by a race car in slip condition, cross-winds and while cornering. The main purpose of the wing is to provide grip by generating downforce during cornering. Research among similar papers to corelate the results with wind tunnel data was used to study the various aspects of the wing and nose section. Surface pressures, symmetricity and wake generation, etc were studied through the CFD analysis using the K-ω turbulence model. The study was carried out for five different yaw angles 0, 6, 8, 10- & 12-degrees angle. It was found that there was asymmetric loading of the wing and a reduction in downforce. A formula-1 car's aerodynamic characteristics operating under yaw is established.

Influence of varying the diffuser angle in the underbody flow analysis is also carried out. Effect of underbody flow at the rear end of the car to generate downforce in seven different diffuser angles is studied and corelated with actual data from wind tunnel.

Key Words: Cornering, Computational Fluid Dynamics (CFD), Slip Angle, K-ω turbulence, Downforce, Lift, Ride Height, Diffuser.

Introduction

The use of wings to generate negative lift or downforce has also been nothing new. The use of wings with very low ride heights has been explored. Ride height changes can bring about drastic change in the results, therefore a ride height of 15mm from the ground is assumed to be constant throughout the study. In the past there has been several researches done on the wing and the nose section to develop the most efficient configuration.

A race car requires maximum grip and hence downforce when undergoing cornering, this is the yawing of the car. Yawing is experienced by a car because of side slip, crosswinds or while cornering.

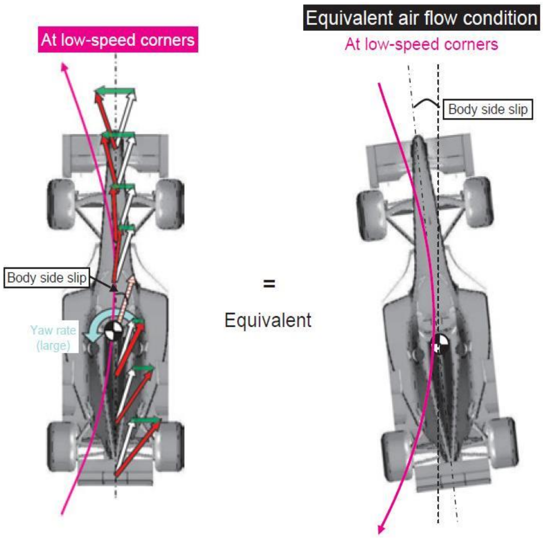

Figure 1: Effect of Side Slip in Racing Car

With this intention of attaining the information on behaviour of the front section of the car, this analysis is carried out.

For the rear of the car, underbody flow and thereof downforce generation is studied and corelated with real time data from wind tunnel for seven different diffuser angles. The bluff body has 32 channels with 2 of the channels dedicated for the atmospheric probe, essentially making it 30 channel probes in the bluffbody.

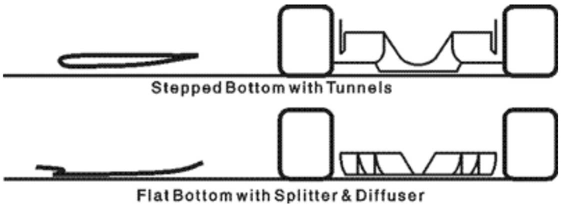

Figure 2: diffuser in race car

Aim and Objective

The aim of this study is to investigate how the vehicle's aerodynamic behaviour changes under yaw of front wing and nose section and rake angles in diffuser.

OBJECTIVE

- Generate a CAD model representing the front wing and nose section of formula-1 car.
- Study the aerodynamic behaviour of the generated CAD in five different yawing angles.
- Resembling the Ahmed body of the bluffbody diffuser used in wind tunnel and understanding the effect of different rake angles in CFD.
- Corelating the data of Ahmed body in CFD with wind tunnel data.

Methodology

A wing and nose section are generated in CAD very similar to that found in Formula-1. Design software Solidworks is used for the same. The bluffbody CAD is made in star ccm+ incorporated in the flowdomain.

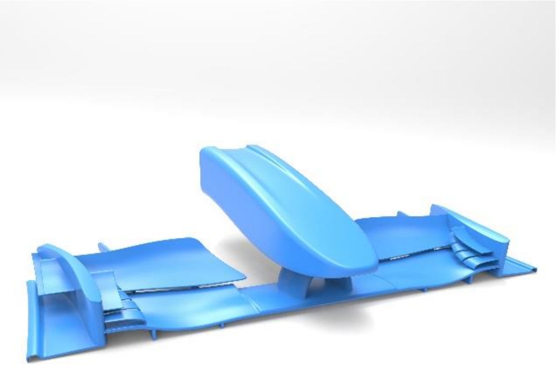

Figure 3: Front Wing-and-Nose section of Formula-1 car in Solidworks

The model is simplified from the actual wings used in formula-1, wherein the number of flaps and cross vanes is lot more than in this model.

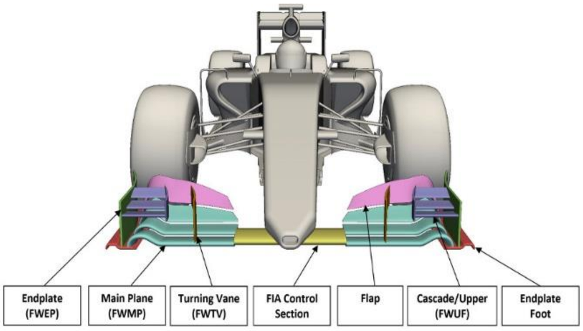

Figure 4: Front Wing Nomenclature of Formula-1 Car

The front wings in the formula-1 car seen above uses complex 3-D geometry which is predominantly designed to manage yaw and interaction with front wheels. From the real design several small changes have been made to the geometry. The number of flaps has been reduced to just two. The upper cascades have been made simpler so are the endplates.

Since, yaw characteristics are studied it is essential to perform full body analysis rather than half analysis. The analysis was primarily done in 3 stages.

STAGE-1

Stage-1 involves the CAD import and optimisation in the simulation software Star CCM+. The CAD model was simplified to reduce computing time and complexity. Flow Domain, which effectively acts as the tunnel was made across the body and made sure that it was sufficiently big, to eliminate blockage.

STAGE-2

Simulations of the wing-nose section at following vehicle speeds and yaw angles have been made.

Table 1: Yaw Angles

CASE	VELOCITY	YAW ANGLE
1	25m/s	0^o
2	25m/s	6^o
3	25m/s	8^o
4	25m/s	10^o
5	25m/s	12^o

These angles are altered by the use of design parameter feature of star ccm. Design parameter helps in controlling the rotation angle of the wing-nose section with respect to an arbitrary axis.

STAGE-3

The next stage is pre-processing before the analysis can begin. In pre-processing, the model is meshed, physics conditions are set and the analysis is ran.

Tyre rotation has important influence in the flow of air across the car especially side pods. Even though tyres are essential to understand the flow across the rest of the car and side pods, it is not been considered for this analysis.

MESH

Surface remesher, trimmed cell and prism layer mesh models are applied to reduce computational time. Subtract feature is used to carry out the analysis. More than 1 million cells and 3.5 million faces are created. Base size of 0.05m is used achieving surface absolute size of 0.0325m.

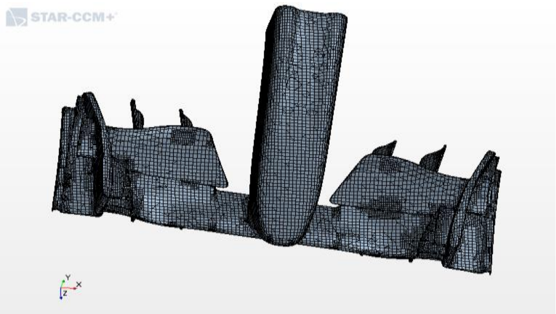

Figure 5: Meshed body using Subtract Feature

A finer mesh is desired as the front wing has lot of small components. Surface and volume control have been added to refine the mesh in the areas needed.

Simulation is run for 1000 iterations and convergence of 1E-03 is achieved for all the simulations.

Results and Conclusions

Drag force, lift force and drag coefficients (C_d) are analysed in this study. The results obtained from the CFD analysis is given below.

Table 2: Derivation in from analysis

ANGLE	LIFT(N)	DRAG(N)	C_d
0^o	-453.2	128.5	0.322
6^o	-523.7	139.9	0.345
8^o	-505.25	141.3	0.350
10^o	-492.5	143.23	0.359
12^o	-449.21	145.8	0.364

Susceptibility of mesh dependant study to dimension less y+ distance. Model is analysed better on the k-ω turbulence solver as gathered from published contents.

It was found that the transition behaviour is better analysed when Y+ value is less than one. Hence a large number of prism layer elements is obligatory to obtain Y+ values less than one. A steady-state, incompressible, segregated flow with k-ω turbulence model are the physics condition.

6 DEGREES YAW

Drag and lift force measurements are studied in through this analysis. It is found that there is decrease in downforce with increase in yaw angle. However, it can be seen from the data in

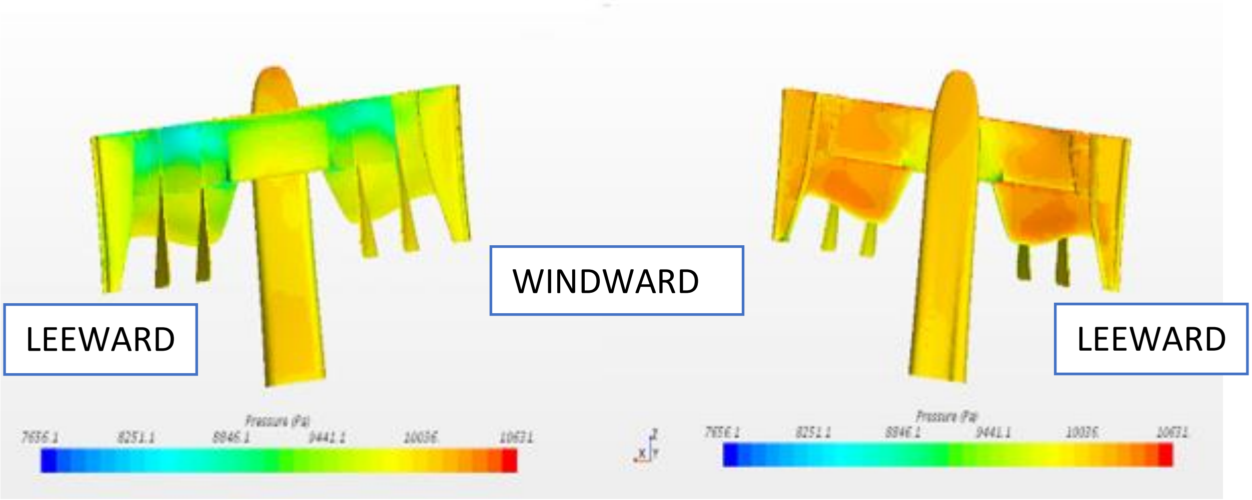

table 2 that the downforce value is highest at 6-degree yaw. The wing is designed to generate best performance at 6-degree yaw. Hence, the downforce value obtained through CFD analysis is justified.

It can be seen from figure 6 that the leeward side generates less pressure and hence creates more suction. Asymmetric loading of surface pressure on the top side can also be clearly observed.

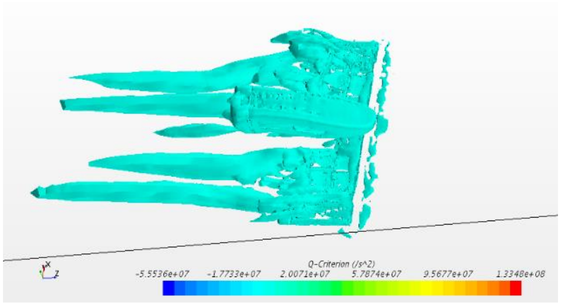

Figure 7: Iso-Surface Q-Criterion for 6 Degree Yaw

The formation of vortices on the lower inside edge and the upper outside edge of the endplate, show similarity to those observed in the previous studies. The lower vortex is formed as the higher-pressure freestream flow moves laterally towards the main-plane suction surface while the upper vortex results because the air moves away from the higher pressure on the flap's upper surface, over the endplate, to the lower freestream pressure.

The development of vortices is shown in figure 7. The iso surface Q-criterion where Q is 2000 s^{-2} is plotted in figure 7. When the pressure is lower than the ambient pressure and Q is positive, vortex is formed. This vortex formed is defined by Q-criterion.

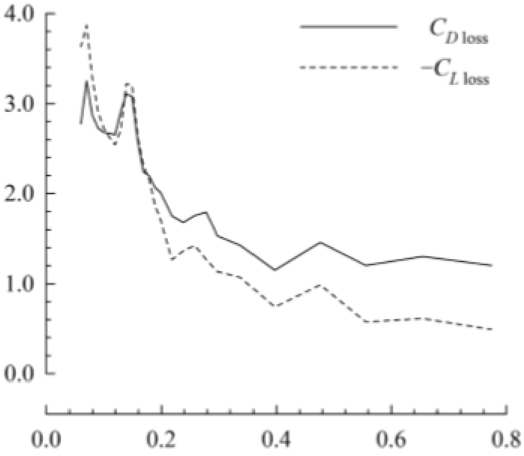

Figure 8: Percentage Loss at various yaw angles and ground clearances as obtained from Roberts Et Al.

Vortices generated by the wings under 6 degrees yaw and under 12 degrees yaw is compared. The difference is quite visible as can be seen below.

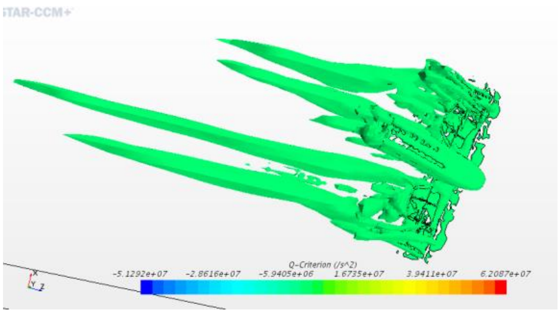

Figure 9: Vortex generation, Iso surface for 12 deg Yaw

It was founded that as yaw was applied, there was significant effect on windward tip vortex.

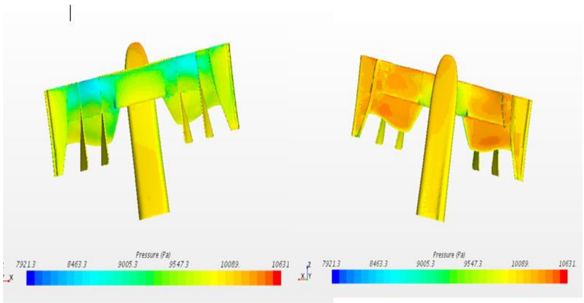

Figure 10: bottom and upside of the wing-nose section under 8-Degree Yaw

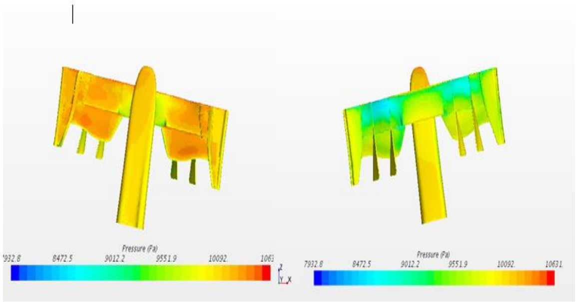

Figure 11: Upside and Bottom of the wing-nose under 10-degree Yaw

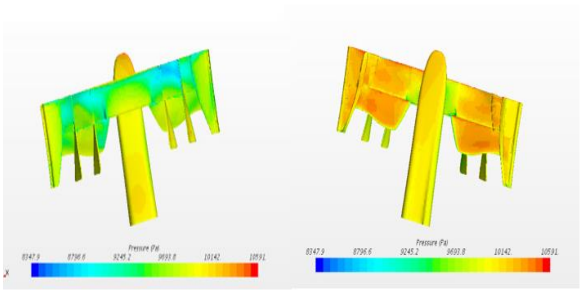

Figure 12: bottom and Upside of the wing-nose section under 12-degree Yaw

Effect of its stability, however, could not be studied with flow visualisation. A region of flow separation appears on the suction surface of the windward flap. This happens because of flow separation and shift in stagnation point from front side of endplate to the edge.

Vortex formation is indicated by the low-pressure regions on the end plate and wing tip. This indicates that flow is circulating from high pressure regions to low pressure regions. Since, we have already observed a change in surface pressure in these regions, the strength of the vortices changes under yaw.

BLUFFODY DIFFUSER

Bluffbody diffuser analysis was carried out to study air flow at the rear end of the car. The diffuser was checked for a seven different rake angles and the CFD data was compared with the real time wind tunnel data for the same ride height.

Probe locations were identified and measured and kept constant for both wind tunnel and CFD analysis. The lateral probe locations on the wind tunnel were replaced by planes in the CFD analysis, for ease of data analysis.

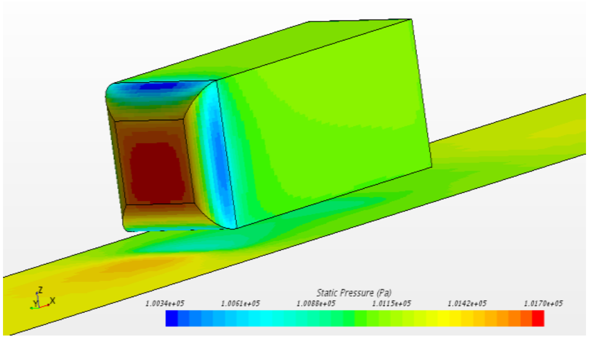

Figure 13: bluffbody 5-degree diffuser angle

A comparison is drawn for all the seven diffuser angles and a correlation has been drawn and represented in graphical manner.

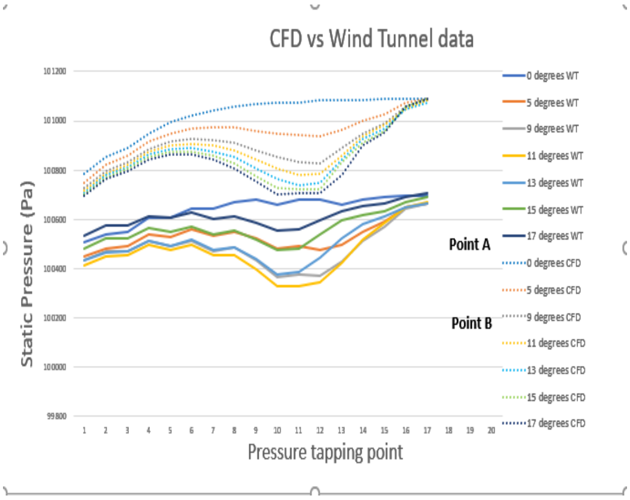

Figure 14: Static Pressure CFD and Wind Tunnel Comparison

It was found that there exists a small difference between CFD data and the wind tunnel data. The reason behind this could be that the wind tunnel flow domain is smaller than in the CFD analysis. This in turn could lead to some blockage on the side of flow domain in wind tunnel. Any such blockage is not possible in computational analysis as the flow domain was made to be sufficiently big to remove any blockage effect.

CONCLUSION

When a car corners, it is essentially cutting through the air in yawed condition. And, while there has been several studies of wings and nose, there is lack of information on the aerodynamic characteristics under yawing condition. Downforce is primarily required in the corners and not in straight lines. It is pretty normal for the teams to look into the stability and sensitivity of the car under pitch, heave and yaw as well. The aim of this study was to investigate the aerodynamics characteristics tests performed by the teams but never published because of high level of competency.

In reality, the race car would experience a maximum yaw of 6 degree after which the effect would die down to insignificance. A similar design to what the formula-1 teams use shows us that the wing has been designed to tackle the real experienced yaw angles. It was found that yaw angles more than that caused a loss of downforce generation and increased drag. A change in stagnation location leads to reduction in downforce, causing a reduction in mass flow under the wing. Although the flow structure changes and pressure differences were not massive, it has to be accounted that this is the level of competency in which these cars perform ad could be the difference between win and lose.

It is important that a racing car generates downforce during cornering, wherein it will have some yaw angle. This study was focussed on that basis to establish a baseline from where complex models and decisions made by the formula-1 teams could be understood.

Bluffbody diffuser correlation was carried out. It was established that due to flow domain differences the correlation was error prone and to get better flow analysis it is imperative to use bigger wind tunnels and bigger scale models. The difference in results are however not too huge and are still comparable.

References

Ahmed, M. (2005). Aerodynamics of a Cambered Airfoil in Ground Effect. *International Journal of Fluid Mechanics Research*, 32(2), pp.157-183.

katz, j. (1995). *Race Car Aerodynamics*. Bentley publishers.

Li, S., Zhou, D., Zhang, Y. and Qu, Q. (2015). Aerodynamic Investigation of RAE2822 Airfoil in Ground Effect. *Procedia Engineering*, 126, pp.174-178.

Roberts, L., Correia, J., Finnis, M. and Knowles, K. (2015). Aerodynamic characteristics of a wing-and-flap configuration in ground effect and yaw. *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, 230(6), pp.841-854.

Lau, C. and Srigrarom, S. (2010). Flow field around the front wing of Formula One racing car model: BAR Honda 003 and MP4-21 underground effect. *International Journal of Aerodynamics*, 1(1), p.72.

Gogel, D. and Sakurai, H. (2006). The effects of end plates on downforce in yaw. *SAE*, SAE paper 2006-01-3647.

BSc, J. and Dominy, R. (1984). Aerodynamic Influences on the Performance of the Grand Prix Racing Car. *Proceedings of the Institution of Mechanical Engineers, Part D: Transport Engineering*, 198(2), pp.87-93.

Panchal, K. Computational Assessment of Front and Rear Wing of Amity Racing Vehicle. (2015). *SAE*, SAE Technical Paper 2015-26-0185.

Soso, M. (2005). *An investigation into the aerodynamics of a wing in ground effect in generic racing car wake flows*. Southampton: University of Southampton.

Zhang, X., Toet, W. and Zerihan, J. (2006). Ground Effect Aerodynamics of Race Cars. *Applied Mechanics Reviews*, 59(1), p.33.