Journal of Material Science and Mechanical Engineering (JMSME)
p-ISSN: 2393-9095; e-ISSN: 2393-9109; Volume 6, Issue 4; July-September, 2019, pp. 245-250

http://www.krishisanskriti.org/Publication.html

CFD Simulation of the Rear Wing of a formula Race Car using Fluent

Nirupam Rohatgi

Faculty, Dept. of Mech. Engg.,
Malaviya National Institute of Technology,
Jaipur, India
E-mail: nrohatgi.mech@mnit.ac.in

Abstract—*CFD simulation of a rear wing of a formula race car has been done using standard k-ε viscous model of Fluent. The results of the simulation have been compared with the results obtained by STAR-CD in the literature. Subsequently, CFD simulation of the rear wing has been done for the speeds ranging from 40 m/s to 67 m/s (90 – 150 mph)and angle of attack (AOA) varying from -4^o to 12^o. Velocity and pressure distribution plots along the surfaces of the aerofoil are presented. Aerodynamic characteristics such as lift and drag coefficients for various speeds have been studied. It was observed that stalling begins to occur at AOA 12^o irrespective of the speed. From AOA -4^o to 4^o, there is an increase of about 35% in Coefficient of lift(Cl) while it remains almost the same from 4^o to 12^o for all the three speeds. Coefficient of drag (Cd) remains almost the same from -4^0 to 4^0 and increases sharply after 4^0. It was found that at a speed of 67 m/s (150 mph) and AOA of 12^0, the coefficient of lift(Cl) has maximum value with a minimum value of the coefficient of drag (Cd). Further, it was observed that the selected wing can be operated within a speed range of 40 - 67 m/s (90 – 150 mph) and AOA varying from 4^0 to 12^{0}with good downward thrust without incurring significant drag losses. There is reasonable agreement between the simulations carried out by STAR-CD and Fluent, however, the difference between the two results suggests that there is a need for experimental validation of the results.*

Keywords—*CFD, Fluent, k-e viscous model, rear wing of a racing car, angle of attack, aerofoil.*

Nomenclature-

- *C* — *Chord length of the aerofoil*
- *Cl* — *Coefficient of lift*
- *Cd* — *Coefficient of drag*
- *AOA* — *Angle of attack*

1. INTRODUCTION

CFD is as an important tool in the field of aerodynamics. It is being widely used to study the aerodynamic design of vehicles. In the case of sports cars, it is used to study the shape of the front and rear spoilers and wings and their effect on the performance of the cars.

A number of researchers have studied rear wings and spoilers of racing cars using CFD. The analysis of a detached spoiler, as per Xu [1] is more complicated than a conventional spoiler. He developed a vortex method based on the no-slip condition for simulating unsteady separated flows around an aerofoil with a detached spoiler [1]. Xu [2] developed two new methods satisfying Kutta condition, incorporated in surface vortices method for solving the steady flow around the aerofoil and unsteady separated flow past an aerofoil with a spoiler.

The CFD simulation of a Hypersonic Cruise Vehicle (HCV) is done in [3]. The design goal of a hypersonic cruise vehicle was to cover 800 km within six minutes with minimum fuel consumption and thermal heating [3]. The simulation was conducted for Mach numbers 0.8, 2,3,5,6,8,10 and angle of attack (AOA) -2^0, 0^0, 2^0, 4^0, 5^0, and 6^0. The results were presented as plots of the coefficient of lift as a function of AOA for various Mach numbers. The strategy for mesh generation for ailerons and spoilers is addressed in [4]. Some of the samples are discussed with varying spoiler and aileron deflections, and Reynolds numbers.

The detailed analysis of the response of various body parts of a car to windy conditions is done by Howell in [5]. The analysis was done for a Rover 800 saloon car, and pressure distributions and yawing moments along the surface of the car were plotted and presented graphically. Van Dam [6] describes recent experiences and practices used in CFD based drag prediction.

The CFD simulation of front and rear wings of Formula race car is done by conducted Armbya, Moujaes, and Kieffer using Star CD simulation code[7].

2. PHYSICAL MODEL

The cross-section of the rear wing examined in this study is shown in Figure 1. The wing has a chord of 17.75 in. (451 mm) (SCCA, Inc). The wing operates at Re = 1.24 × 10^6 at 40 m/sec (90 mph) and at Re = 1.65 × 10^6 at 54 m/sec (120 mph) [7].

In case of racing-cars the wings are used in an inverted position so as to create downward force instead of the lift.

Accordingly, the angle of attack nomenclature is reversed of the aerodynamic convention as applied to an aircraft.

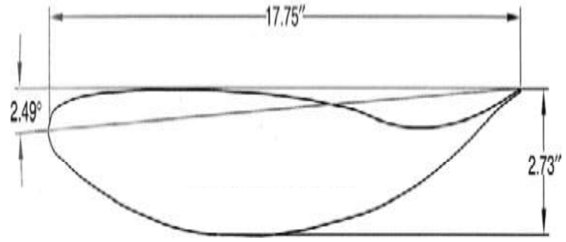

Figure 1: Geometry of the rear wing studied[7]

A positive angle of attack means that the leading edge is lower as compared to the trailing edge. The rear wing was studied at -4^0, 0^0, 4^0, 8^0, 12^0, and 16^0 AOA. The AOA of 16^0 was chosen as the maximum angle of attack, and on the opposite side, AOA of -4^0 was chosen as the maximum angle of attack.

3. COMPUTER MODEL

The profile of the aerofoil used for the study was the same as the one adopted by Armbya, Moujaes and Kieffer [7]. The problem was formulated as a two-dimensional problem, considering uniformity in the third direction. The computational domain around the aerofoil was selected such that the leading edge of the aerofoil is located at a distance of 12.5 times the chord length, 'C' from the inlet boundary, and at a distance of 12.5 times 'C' from the upper and the lower free stream boundaries. Outlet boundary was taken at a distance of 20 times 'C' from the trailing edge, as recommended by C.P. Van Dam [6].

Geometric model of the rear wing was created in GAMBIT. The total number of cells in the grid were 12,285. The grid adopted is shown in Figure 2.

Race-cars have a finite depth of aerofoil, however 2-d simulation has been done due to the limitation of computational resources. The present analysis captures the main features of the aerofoil geometry and the effect of the important parameters such as the angle of attack and velocity. The end effects, however, need to be considered for more accurate analysis using a 3-d analysis.

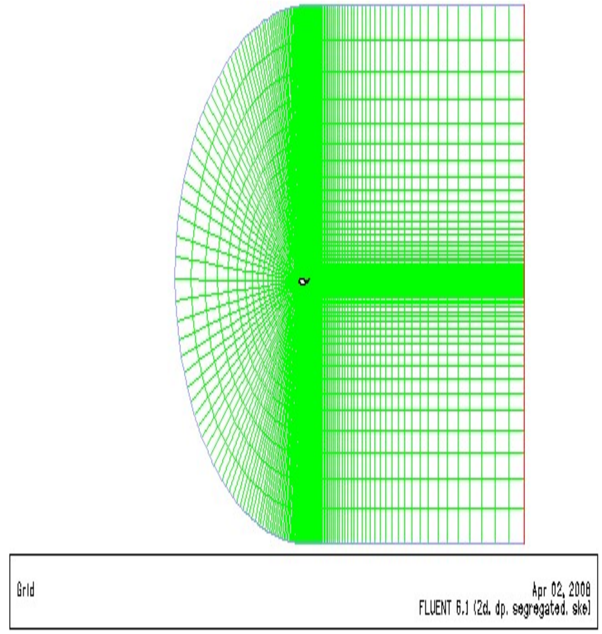

Figure 2: Grid pattern used for simulation by Fluent

2-D simulation has been done in FLUENT using standard k-ε viscous model. The flow was assumed to be isothermal and the energy equation was not activated.

The angle of attack was changed by changing the velocity vectors. The discussion about turbulent model are done in [9].

3.1 Input parameters and Constants

The input values used for the simulation were standard sea level values for the free stream velocities, which are:

Pressure = 101,325 Pa,

Density = 1.2250 Kg/m^3,

Temperature = 288.16 K, and

Kinematic Viscosity = 1.4607e-5 m^2/s.

The energy equation was not activated as the flow was assumed to be isothermal and adiabatic.

4. VALIDATION OF RESULTS

Validation of the results has been doneat a speed of 52 m/s.

The simulation was performed for different AOAs. The variation of aerodynamic parameters, namely Coefficient of lift (Cl) and Coefficient of drag (Cd) with AOA is shown graphically along with that obtained using STAR-CD by Armbya, Moujaes and Kieffer [7].

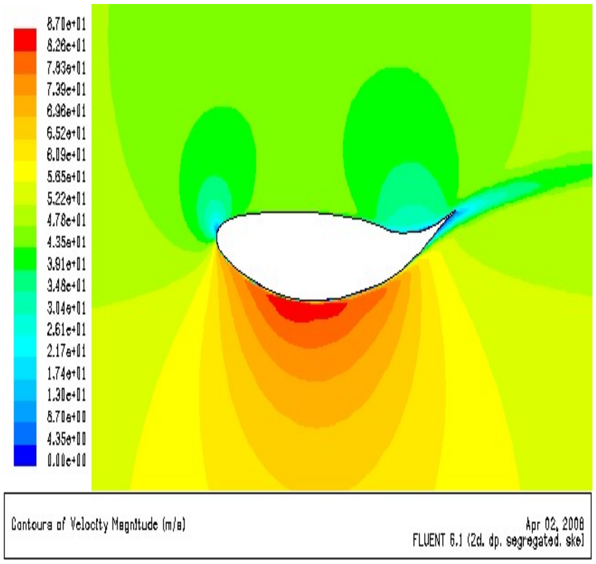

Figure 3. Velocity contour for 0^0 AOA by Fluent

For the flow velocity of 52 m/s and AOA 0^0,Figure 3 shows the velocity contours with a maximum velocity of 87 m/s reached at the bottom of the aerofoil. As expected, the magnitude of velocity on the bottom side of the spoiler is much greater than that at the topside resulting in high downward force. Similar results obtained by[7] is shown in Figure 4. From the two plots, it can be seen that there is a difference in the maximum velocity between the two results, but the velocity profiles remain more or less the same in both the results.

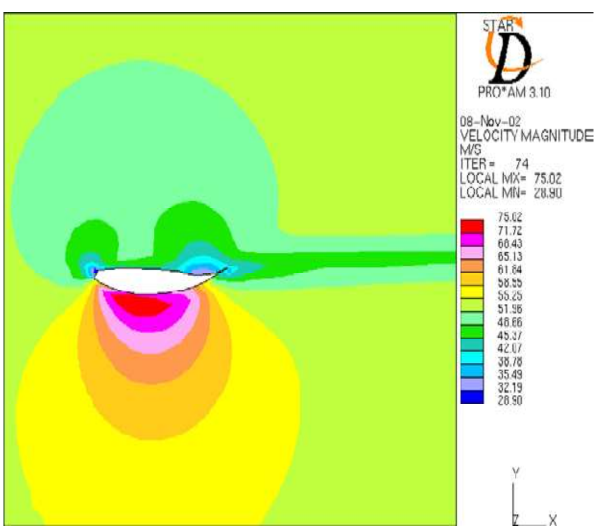

Figure 4: Velocity contour for AOA of $0^0$from [7]

The variation in pressure coefficient is shown in Figure 5 and Figure 6. There is a significant difference in pressure coefficient as obtained by Fluent and that obtained by [7], both in magnitude and in shape.

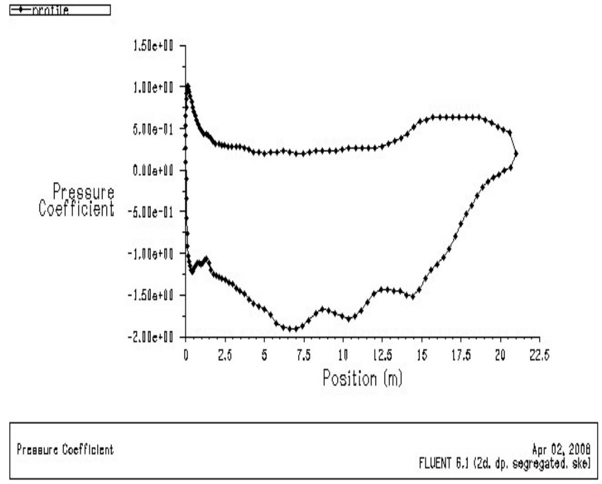

Figure 5. Pressure coefficient for AOA of 0^0 by Fluent

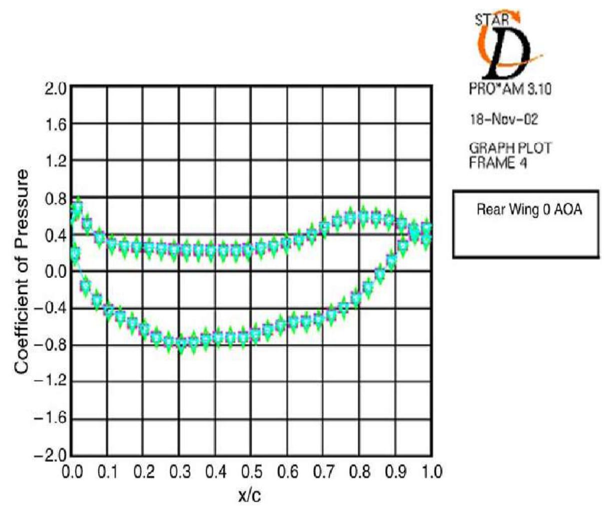

Figure 6. Pressure coefficient for AOA of $0^0$from [7]

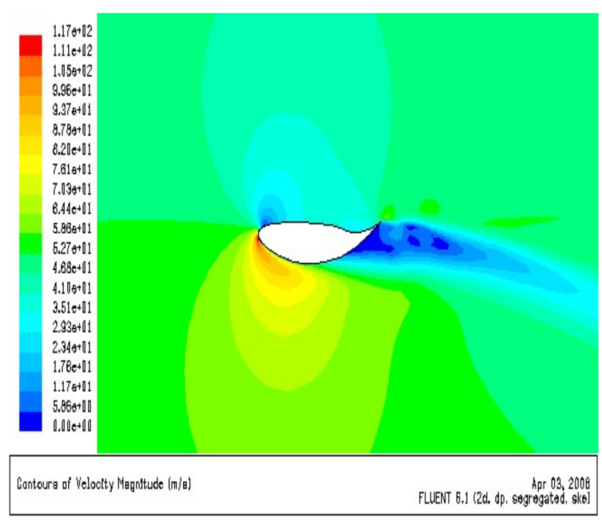

Figure 7: Velocity contour for AOA of 16^0 by Fluent

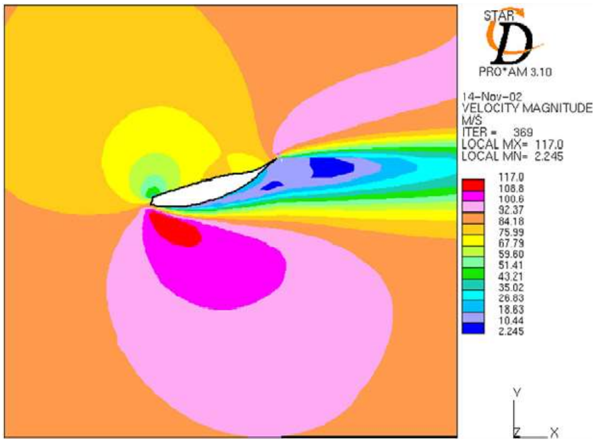

Figure 8: Velocity contour for 160 AOA from [7]

Similarly, the velocity contours at AOA of 16^0 for the two have been shown in Figure 7 and Figure 8. The occurrence of stalling is seen in both the figures, indicating a substantial similarity. Even the velocity profile and magnitude of maximum velocity is the same in both the figures.

The plots of Cl and Cd with AOA as obtained by the simulation and by Armbya, Moujaes, and Kieffer[7]are shown in Figure 9 and Figure 10 respectively. There is a considerable difference in the value of Cl, but the Cd remains almost the same.

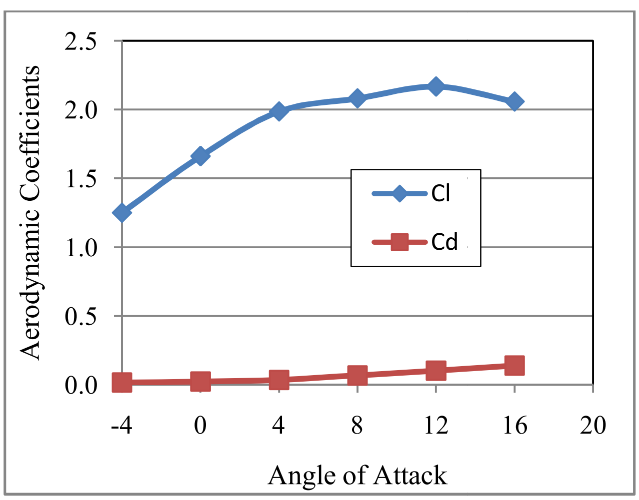

Figure 9. Variation of Cl and Cd as a function of AOA by Fluent

The variations in Cl and Cd show some similarity. Stalling in both the cases occurs at AOA 12^0, and the decrease in Cl, of about 10% after the stalling, is also the same in both the cases.

From the above discussion, it can be seen that thereare similarities as well as differences in both cases. The similarities are natural and expected. The reason for differences seems to depend primarily on the software adopted for the simulation. Therefore, the results both by Fluent and STAR-CD need to be further evaluated and validated with the experimental results as both the results seem realistic.

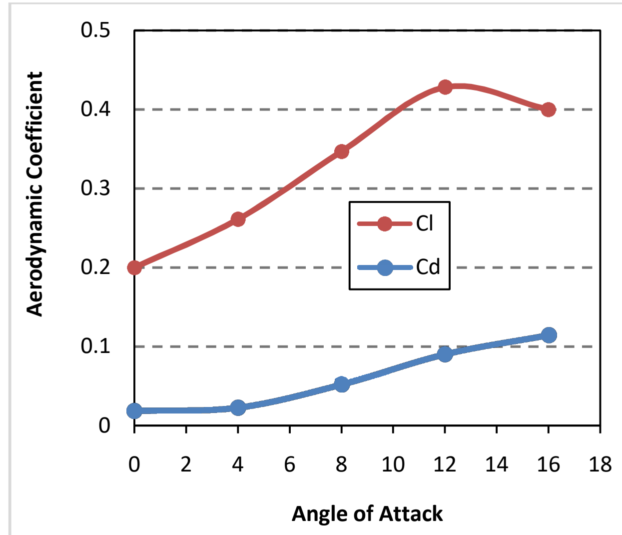

Figure 10. Variation of Cl and Cd as a function of AOA from [7]

5. RESULTS AND DISCUSSIONS

For a flow velocity of 52m/s and AOA of 0^0, the maximum velocity of 87 m/s was obtained at the bottom of the aerofoil. The magnitude of the velocity on the bottom side of the spoiler is much greater than that at the topside resulting in high downward force. Similar results were obtained by Armbya, Moujaes, and Kieffer[7].

Figure 11 and 12 show variation of Coefficient of lift (Cl) and coefficient of drag (Cd) with the angle of attack (AOA).

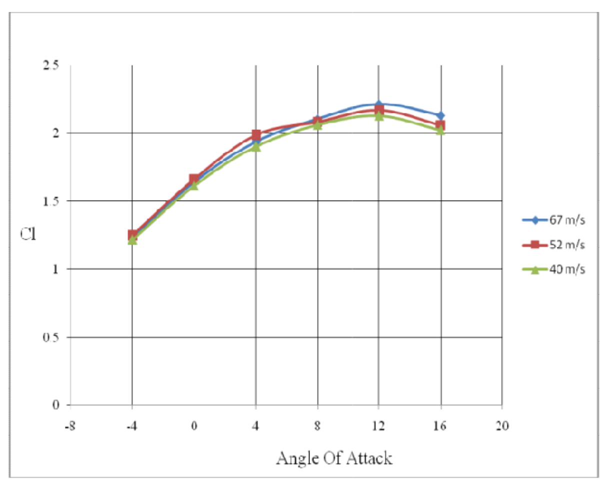

Figure 11: Coefficient of lift (Cl) with AOA by Fluent

From the above graphs, it can be inferred that stalling occurs at AOA of 12^0 irrespective of the speed. Further, there is only a marginal difference in the value of Cl for all three speeds.

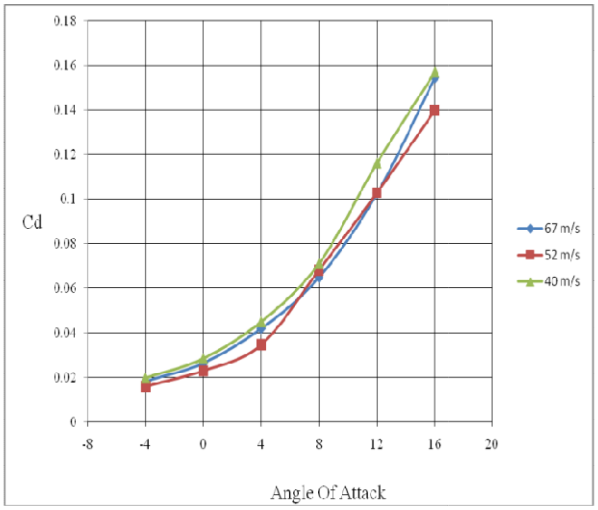

Figure 12: Coefficient of drag (Cd) with AOA by Fluent

The combined plot of pressure coefficient for all the three speeds is given in Figure 13.

The values in pressure coefficient for all three speeds hardly have any difference, indicating that the difference in pressure at the top and bottom side almost remain constant irrespective of the speed. The marginal difference in the value of Cl also points to the same conclusion.

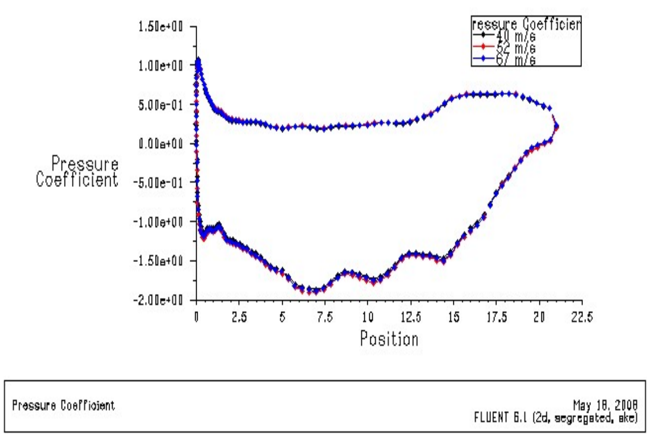

Figure 13. Combined plot of pressure coefficient by Fluent

The maximum value of Cl and the corresponding value of Cd at different speeds are given in table 1.

From the table 1, it is observed that at a flow speed of 67 m/s(150 mph)and at AOA of 12^0, the value of Cl is maximum while corresponding value of Cd is minimum.

Table 1: Maximum Value of Cl with corresponding value of Cd for different speeds by Fluent

Speed (m/s)	AOA	Cl	Cd
40	12^0	2.1280	0.1163
52	12^0	2.1674	0.1029
67	12^0	2.2138	0.1025

6. CONCLUSIONS

A two-dimensional CFD study has been performed on the aerofoil profile of rear wing of a race car for different AOA and speeds using Fluent. Detailed velocity and pressure variation plots along the surfaces of the aerofoil have been presented.

The validation of results with published results [7] has been done. While the value of Cd, velocity and pressure profiles are in agreement with [7], the magnitude of maximum velocities and value of Cl happen to be substantially higher than that obtained from [7]. The immediate reason for the difference is that the two different software's have been used to study the rear wing. Since both the results seem to be realistic, it is suggested that the results shall be validated experimentally.

In addition to the validation at 52 m/s, simulations were performed for 40 m/s and 67 m/s also. The results showed that stalling occurs at AOA 12^0 for all three speeds. From 12^o to 16^o, the value of Cl falls by about 10% due to stalling. It was also observed from Figure 11, that in the range of AOA 4^0 to 12^0, the difference in the value of Cl is marginal (i.e. 5%) for all the three speeds while for AOA varying from 4^0 to -4^0, the value of Cl falls by about 35%.

Cd remains almost stagnant between AOA of -4^0 to 4^0 and increases sharply by about 150% from AOA 4^o to 12^0. From AOA 12^0to16^0, Cd continues to increase steeply. Hence, the best range of AOA for all three speeds is 4^0 to 12^0. Also, for low speeds, the value of Cl required is low. Hence the vehicle can be satisfactorily operated at a low angle of attack, i.e. nearer to 4^o thereby reducing undesired drag on the vehicle or Cd.

For the aerofoil selected, the aerodynamic performance is significantly affected by AOA while there is a marginal effect of speed on Cl and Cd. The results can help in better design of rear wing of racing cars.

7. FUTURE WORK

For future work, it is suggested to perform simultaneous study by altering the profile, AOA and speed in a more elaborate manner so as to obtain optimum performance characteristics at the desired speed. The effect of the front wing can also be analyzed and validated, which was not done here. Also, the study can be further extended to the next level by introducing the energy equation, thereby studying the temperature gradients around the race car and its effect on aerodynamic coefficients. Above all CFD simulation need to be validated experimentally as the two software's differ in their results at some points.

REFERENCES

[1] Xu, C. (1999). A vortex method for separated flow around an airfoil with a detached spoiler. *Computational Mechanics* , 271-278.

[2] Xu, C. (1998). Surface vorticity modeling of flow around airfoils. *Computational Mechanics* , 526-532.

[3] Feng, D., Yu, Y., & Wang, X. (2007). Modeling Method and CFD Simulation of a Hypersonic Cruise Vehicle. *Second IEEE Conference on Industrial Electronics and Applications* , 1346-1349.

[4] Colak, B., Elsholz, E., Mertins, R., & Barakat, S. (2005). 3D viscous flow analysis on wing-body-aileron-spoiler configurations. *Aerospace Science and Technology* , 476-484.

[5] Howell, J. (1996). The side load distribution on a Rover 800 saloon car under crosswind conditions. *Journal of Wind Engineering and Industrial Aerodynamics* , 139-153.

[6]Van Dam, C. P., (1999). Recent experience with different methods of drag prediction. *Progress in Aerospace Sciences* , 751-798.

[7]Armbya, N., Moujaes, S., & Kieffer, W. (2006). CFD study of section characteristics of Formula Mazda race car wings. *Mathematical and Computer Modelling* , 1275-1287.

[8] Mehta, F. (2005-2006). CFD Analysis of a Formula One front wing. *M.Sc. Thesis, Cranfield University* .

[9] Fluent Inc. Fluent user's guide. Version 6.21, Lebanon, NH, USA: Fluent Inc.; 1998