International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622
National Conference on Advances in Engineering and Technology
(AET- 29th March 2014)

RESEARCH ARTICLE OPEN ACCESS

Numerical Simulation of Drag Reduction in Formula One Cars

A.Muthuvel *, N. Prakash **, J. Godwin John***
*(Department of Automobile Engineering, Hindustan University, Chennai-603103
Email: muthuvelcfd@gmail.com)
** (Department of Automobile Engineering, Hindustan University, Chennai-603103
Email: nprakrao1@gmail.com)
** (Department of Automobile Engineering, Hindustan University, Chennai-603103
Email: godwinjohn18@gmail.com)

ABSTRACT
Today, it is very usual to see numerous cars, from commercial cars to sports cars fitted with different types of spoilers on them. The exterior fashioning and aerodynamically well-organized design for reduction of engine load which reflects in the reduction of fuel consumption and producing the down force for the stability are the two essential factors for an effective operation in the modest world. The adding of rear spoiler to an aerodynamically optimized car body will result in a change of lift and drag forces the car experiences and thus influence the cars overall performance, fuel consumption, safety, and stability. This paper presents a discussion on the results obtained from numerical simulation of airflow over a F1 car for various speeds like 80m/sec, 100m/sec and 120m/sec with and without a rear spoiler for 0^0 and 5^0 angle of attack of the spoiler. The influence of rear spoiler on the generated lift, drag, and pressure distributions are investigated and reported.
Keywords – Aerodynamics, Drag, Fuel consumption, Lift, Spoilers.

I. INTRODUCTION

Flow over body has been a subject of great number of investigation mainly because of wider engineering applications. Some examples are flow over car, buildings, flight-deck of a ship, underwater appended vessels like submarine, torpedo, automated underwater vehicle (AUV), remotely operated vehicle (ROV) etc. In [1] author described the identification of aerodynamic noise source around a coupe passenger car and rear spoiler is added to the vehicle and acoustic effects are investigated. It is found that installing a rear spoiler can change the dominant noise source (location) from front bumper to the rear spoiler. Subsequent to noise source identification, the effect of different angles for rear spoiler is studied in order to recognize the case that gives the minimum acoustic power level of the dominant source (rear spoiler). By increasing the vehicle cruise, the aerodynamic noise rises significantly (e.g. the maximum acoustic power level increases around 3%). Size of the wake formed behind the rear spoiler and the turbulent intensity distribution on it, confirms that there exists a case that generates lower air-born noise. A pressure-based implicit procedure to solve Navier-Stokes equations on a unstructured polyhedral mesh with collocated finite volume formulation is used [2] to simulate flow around the smart and conventional flaps of a spoiler section under the ground effect. The agreement between presented predation and experimental data is for smart flap is smoother than conventional flap. In [3], authors carried out the work for numerical simulation of airflow over a passenger car without a rear spoiler and compares these with results obtained for a passenger car fitted with a rear spoiler and he suggested a rear spoiler on the generated lift, drag, and pressure distributions are investigated and reported. Two different types of simulations are performed [4] for the flow around a simplified high speed passenger car with a rear-spoiler and the other for the flow without a rear-spoiler. The standard k-ε model is selected to numerically simulate the external flow field of the simplified Camry model with or without a rear-spoiler. Through an analysis of the simulation results, a new rear spoiler is designed and it shows a mild reduction of the vehicle aerodynamics drag. This leads to less vehicle fuel consumption on the road. In [5], authors presented a comprehensive study for realistically predicting airflows around cars. The focus is on high fidelity road vehicle simulations, but with as short as possible turnaround time as prerequisite for aerodynamic optimization and innovation at lower development cost. The airflow is modeled using different commercial CFD packages, i.e. Ansys Fluent, CFX, Open FOAM and Power FLOW. Furthermore, recommendations for geometry preparation, grid and case set-up are given. Results for a road vehicle indicate that the best solver from an accuracy point of view is Star-CCM+. In [6] authors described about the drag reduction by checking car models with the installation of external devices and without the devices. After validating it is found that the LES gave the good results. By installing the devices it is found the drag coefficient variation is about average of 18%. These were for tractor trailer. One SUV model was also simulated. A rear wing modeled by NACA 0015 was attached at 10 degrees from the rear slant angle which lowered the drag-by-area of the Ahmed car model by 10%.All results were compared against experimental results. Tractor trailer RANS results showed an error of 12% while LES results showed an error of 4.9% in comparison with the paper's results. The SUV model showed an error of 5.7% in comparison with the experimental results for a small scale Hummer model in a wind tunnel.

A detailed discussion about the drag reduction by means of using rear spoilers is carried out [7]. Adding a spoiler at the very rear of the vehicle makes the air slice longer, gentler slope from the roof to the spoiler, which helps to reduce the flow separation. Reducing flow separation decreases drag, which increases fuel economy. In [8], author highlighted the turbulence areas and how to reduce it and he suggested that real trouble is that the winglets and the lower rear wing element, which are in interaction with the diffuser, produce the most turbulence. The spallartallmaras based DES of Ahmed reference model with the slant angles 25 ° and 35^0 is discussed [9]. At 25 ° RANS gives relative results for it is in good agreement as flow is attached and is no good in the wake region. At 35° slant angle DES is in good agreement with the experimental analysis in the wake region. DES is also good with the drag as in experimental analysis. They suggested that des gives good result in the wake region. **Nari Krishnani**[10]discusses about the drag reduction in the vehicle and checked the feasibility of using the external devices for reducing drag on large size SUV. The study suggests that STARCCM+ software gives the better result than other software and by using the spoiler we will get the less drag force and better stability.

II. ANALYTICAL RESULTS AND ANALYSIS OF F1 CAR

Three dimensional numerical analysis of the flow over F1 car is carried out with the general purpose Reynolds Averaged Navier-Stokes Equations (RANSE) solver STAR-CCM+. Steady analysis is carried out with unstructured polyhedral mesh with AKN k-ε model turbulence models. For all the cases, the maximum residual from continuity, x-momentum, y-momentum and z-momentum is restricted to 1e-4 as convergence criteria. Initially, 300 iterations are carried out with first-order upwind scheme and relaxation factor for velocity as 0.3 and pressure as 0.1 to guard against divergence of the solution. Later, till the convergence criteria are met, iterations are carried out with second-order upwind scheme with velocity relaxation factor as 0.5 and pressure relaxation factor as 0.3 to obtain higher accuracy and also to accelerate the convergence.

III. GEOMETRICAL MODELING

Three dimensional F1 CAD models of front view is shown in Fig.1 its side view is shown in Fig. 2 and top view is shown in Fig.3 by using the Solid works CAD software with its actual dimensions.

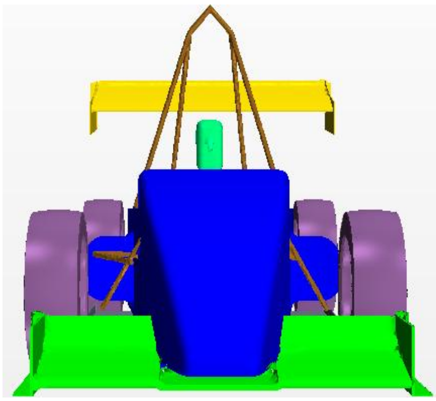

Figure 1: Front View

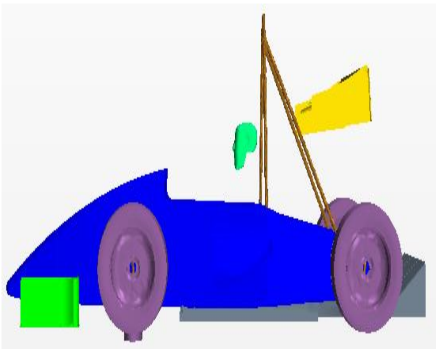

Figure 2: Side View

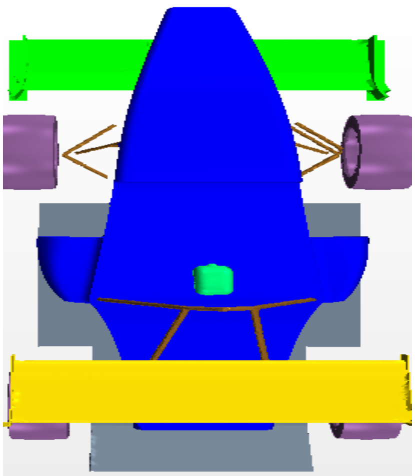

Figure 3: Top View

IV. GRID GENERATION

Unstructured polyhedral mesh is chosen while generating the grid. During generation of the meshes, attention is given for refining the meshes near the F1 car so that the boundary layer can be resolved properly. The typical mesh for car is shown in Fig.4 its sectional view is shown in Fig.5 and a magnified view near the solid wall of car is shown in Fig.6.

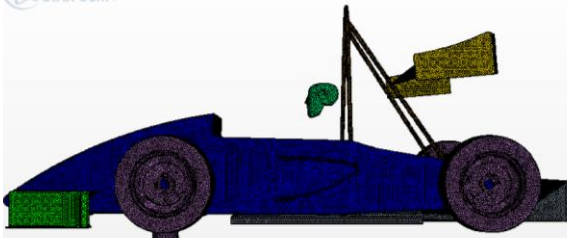

Figure 4: Mesh on Car

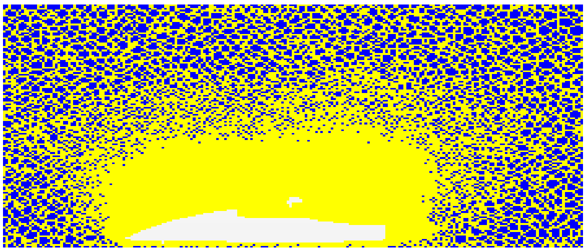

Figure 5: Sectional View of Polyhedral Mesh

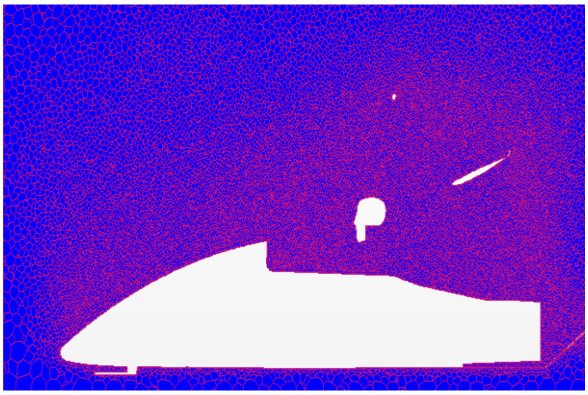

Figure 6: Magnified View of Polyhedral Mesh

V. BOUNDARY CONDITIONS

Boundary conditions were applied on the meshed model using the STARCCM+ CFD software. The analysis was carried out in moving road and rotating wheel condition. In the simulation only straight wind condition was considered at 3 different vehicle speed of 80, 100,120 m/sec. Constant velocity inlet condition was applied at the inlet to replicate the constant wind velocity conditions same as wind tunnel tests. Zero gauge pressure was applied at the outlet with operating pressure as atmospheric pressure. All the boundary conditions used in the analysis are listed in Table 1 and its computational domain for numerical simulation is shown in Fig. 7.

TABLE 1: BOUNDARY CONDITIONS

BOUNDARY	BOUNDARY CONDITIONS	VALUES
Inlet	Constant velocity	V=80,100 & 1200m/sec
Outlet	Pressure outlet	Constant pressure= 0 N/m2
Bus body	No slip – stationary wall	-
Domain	Constant velocity	V=80,100 & 1200m/sec

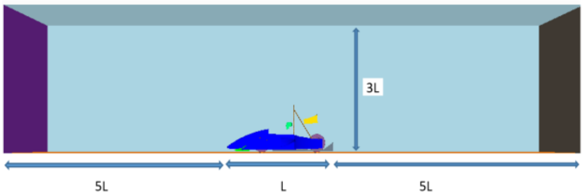

Figure 7: Computational Domain for the F1 Car Analysis

VI. RESULTS FOR UNSTRUCTURED MESH

Grid independent test is carried out for the F1 car for 1.52 million, 2.47million and 3.54 million cells. AKN k-ε two layer models were used for the test. The drag coefficient values along the surface are measured. From Fig. 8 it can be seen that the values at 1.52 million is deviating from the values obtained with 2.47 million cells. However, difference between the values obtained from 2.47 million cells with those 3.54 million cells is very less (less than 5%).Hence, 2.57 million cells is considered for further analysis for F1 car. The CFD analysis of flow over the F1 car is made for various speeds with 0^0 and 5^0 angle of attack for spoiler. Drag force is measured for various speeds like 80,100 and 120m/sec are shown in Table 2 and Table 3.The drag force and normal force are gradually increasing w.r.t to high speed.

Table 2: Results For 0^0 Angle Of Attack

Speed	Pressure Force	Shear Force	Total Axial Force	Pressure Force	Shear Force	Total Normal Force
80	190.85	6.05	196.90	-184.43	0.72	-183.711
100	299.81	9.52	309.33	-306.50	1.28	-305.22
120	422.08	13.24	435.36	-443.44	1.79	-441.69

Table 3: Results For 5^0 Angle Of Attack

Speed	Pressure Force	Shear Force	Total Axial Force	Pressure Force	Shear Force	Total Normal Force
80	196.33	5.75	202.09	-175.01	0.65	-174.39
100	306.38	9.36	315.75	-276.83	1.30	-275.50
120	443.50	13.12	456.60	-373.49	1.72	-371.77

The velocity contour and pressure contour diagram for AKNk-ε two layer model is shown is shown in Fig. 11 and Fig 12 for velocity 80/sec. From the velocity contour and pressure contour diagram, the variation of the velocity and pressure of flow over the F1 car are seen. At the bow of the AUV, stagnation condition is clearly captured. Also, boundary layer formation is seen near the walls. As the flow advances over the body, the velocity increases gradually and then reduces in the rear region due to curvilinear nature of the rear geometry. Similarly for 100 m/sec is shown in Fig. 16-17 and for 120 m/s is shown in Fig. 21-22.The vortices are formed behind the front part and rear of the spoiler is shown in Fig.8, 13 and 19. Pressure and velocity path lines are clearly showing the stagnation points and high stream flow area are in Fig. 9 -10 for 80m/sec, Fig. 14 - 15 for 100m/sec and Fig. 19-20 for 120 m/sec.

Analytical results for 80 m/sec

0 Degree **5 Degree**

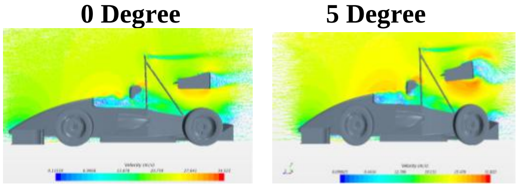

Figure 8: Velocity Vectors on The Body

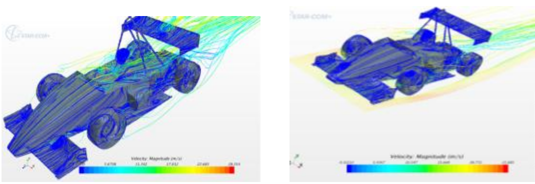

Figure 9: Velocity Path Line on The Body

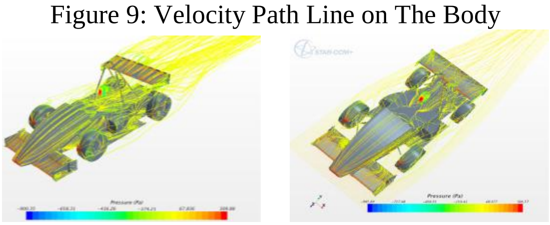

Figure 10: Pressure Path Line on The Body

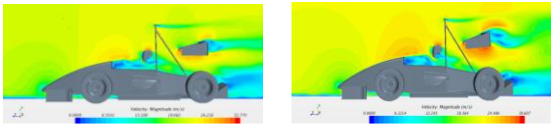

Figure 11: Velcity Contours on The Body

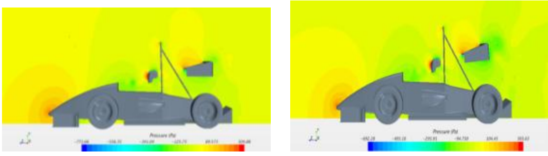

Figure 12: Velcity Contours on The Body

Analytical results for 100 m/sec

0 Degree **5 Degree**

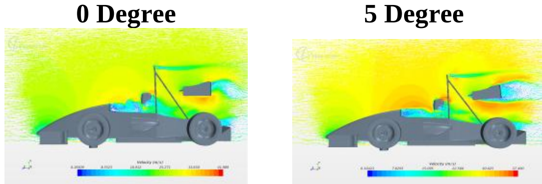

Figure 13: Velocity Vectors on The Body

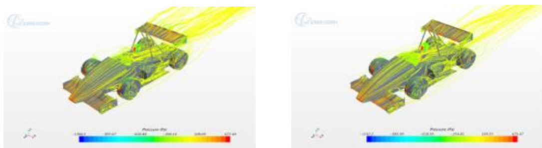

Figure 14: Pressure Path Line on The Body

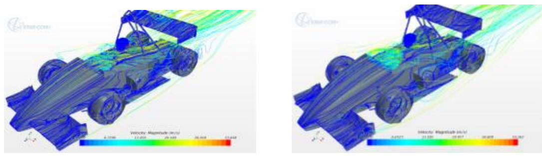

Figure 15: Velocity Path Line on The Body

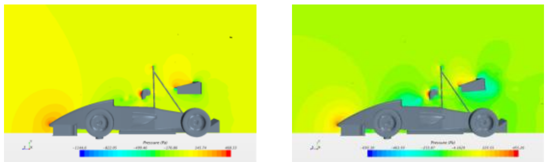

Figure 16: Pressure Contours on The Body

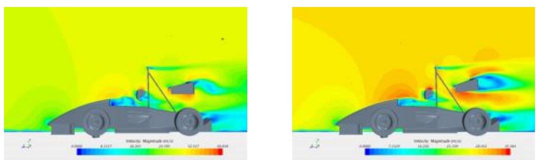

Figure 17: Velcity Contours on The Body

Analytical results for 120 m/sec

0 Degree **5 Degree**

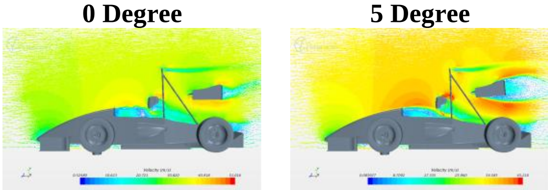

Figure 18: Velocity Vectors On The Body

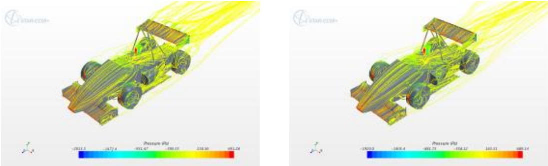

Figure 19: Pressure Path Line On The Body

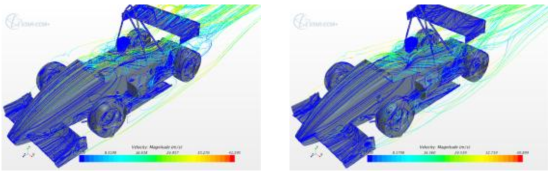

Figure 20: Velocity Path Line On The Body

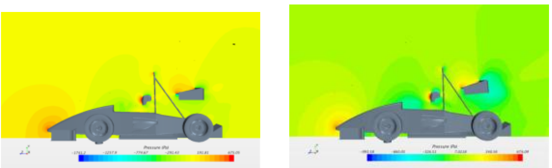

Figure 21: Pressure Contours On The Body

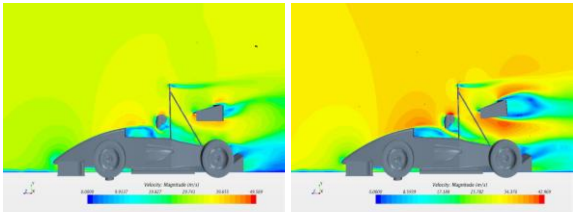

Figure 22: Velcity Contours On The Body

VII. CONCLUSION

In this paper, analysis is carried out for F1 car with commercial code STAR-CCM+. Analysis is carried out for F1 car with spoiler for unstructured to 0^0 and 5^0drift angles of spoiler. In this analysis, AKN k-ε turbulence model is used. The normal and axial forces are obtained from the present analysis.

It is seen that pressure force and shear forces are gradually increasing w.r.t to speed due to this drag and normal forces are increasing for 0^0 and 5^0 spoiler drift angles. For this F1 car at 0^0 spoilers gives less drag then 5^0 spoilers.

REFERENCES

[1] SajjadBeigmoradi, KambizJahani, Arash Keshavarz and Mohsen BayaniKhaknejad, “Aerodynamic Noise Source Identification for a Coupe Passenger Car by Numerical Method Focusing on the Effect of the Rear Spoiler”, SAE International, 2013.

[2] M.H. Djavareshkian, A. Esmaeli, “Optimization of Smart Spoiler for Race Car”, International Conference on Fluid Dynamics and Thermodynamics, Dubai, January 25-27, 2011, 1-8.

[3] Shyam P. Kodali and SrinivasBezavada, “Numerical Simulation Of Air Flow Over A Passenger Car and The Influence of Rear Spoiler Using CFD”, International Journal of Advanced Transport Phenomena, 1(1), 6-12, Jan-Dec, 2012.

[4] Xu-xia Hu, Eric T. T. Wong, “A Numerical Study On Rear-spoiler Of Passenger Vehicle”, World Academy of Science, Engineering and Technology, 57, 636-641, 2011.

[5] ȘtefanBordei, Florin Popescu, “Aerodynamic results for a notchback race car”, The Annals of “Dunărea De Jos”, University of Galați Fascicle, Technologies in Machine Building, ISSN 1221- 4566, 2011.

[6] Walid Ibrahim Mazyan, “Numerical simulations of drag-reducing devices for ground vehicles”, 2013.

[7] Mustafa Cakir, CFD study on aerodynamic effects of a rear wing/spoiler on a passenger vehicle, Dec 2012, Santa Clara University, School of Engineering, 2012, 1-124.

[8] MiladMafi, “Investigation of turbulence created by formula one cars with the aid of numerical fluid dynamics and optimization of overtaking potential”, Ansys Conference and 25th CADFEM’s User Meeting, 21-23 Nov, 2007.

[9] Sagar Kapadia and Subrata Roy, Matthew Vallero, KennethWurtzler and James Forsy, “Detached-eddy simulation over a reference Ahmed car model”, from http://cpdl.kettering.edu/ahmed-dles.pdf.

[10] Pramod Nari Krishnani, “CFD Study of Drag Reduction Of A Generic Sport Utility Vehicle”, 2006.