International Journal of Modern Trends in Engineering and Research
www.ijmter.com
e-ISSN No.:2349-9745, Date: 2-4 July, 2015

Numerical Analysis of a Spoiler Arrangement in Formula Car

Ganesh. Y[1] Prakash. N[2], Muthuvel.A[3], P. Baskar Sethupathi[4]

[1] *Dept of Automobile Engg. Hindustan University, yuvaganeshaero@gmail.com*
[2] *Dept of Automobile Engg. Hindustan University, nprakrao1@gmail.com*
[3] *Dept of Automobile Engg. Hindustan University,muthuvelcfd@gmail.com*
[4]*Dept. of Automobile Engg. SRM University, sethupathi.b@ktr.srmuniv.ac.in*

Abstract - Now a day's formula vehicle becomes very familiar to every one due to peoples having so much crush and passion on Racing and Automobiles. Aerodynamics plays a pulsating role in engine performance and efficiency of the vehicle , for that designer should consider two things in mind, the aerodynamic shape of the vehicle and the aerodynamic force experienced by the vehicle while vehicle in motion. To get maximum performance of the vehicle we need know the aerodynamics forces acting on the vehicle and how to utilize those forces for increasing the performances and stability. In this paper the drag reduction for a formula vehicle is analysed by streamlined flow over the frontal area. In order to achieve the stability and drag reduction spoiler is fixed on the vehicle at rear area for variousangles of attack from -10^0to10^0 and the vehicle is analysed for various drift angles from (0^0to35^0). For stabilizing the vehicle we need to find at which angle the vehicle reduce less drag and more downforce for stability. The reduction in Drag will reduce the fuel consumption, increase vehicle performance like increasing top speed, by this vehicle leads to overtaking during race. Based on the grid independent study numerical simulation was carried out for various turbulence models with unstructured polyhedral mesh. The modelling of vehicle is done by using CATIA V5R18 and the numerical analysis of vehicle is being accomplished by the CFD commercial software (STAR-CCM+).

Keywords: Aerodynamics; Formula Car; CFD; Spoiler; Drift angles; Drag Reduction**.**

I. INTRODUCTION

The formula car is designed in such a way that it will produce only less amount of drag While designing the vehicle the designer should kept two things in mind ,the frontal area should be minimum as possible as. Then adding a rear wing spoiler will produce less amount of drag. A spoiler is an aerodynamic device which is used to develop the negative turbulence for reduction of drag force and increasing the downforce by underneath separation and around the vehicle for increase the stability.

Muthuvel et al. [1] carried out work for numerical simulation airflow over a formula one vehicle, the spoiler is fitted at rear side of the vehicles at different angles 00 and 50. The vehicle is operated at various speeds 80KMPH, 100 KMPH and 120KMPH. Analysis is carried out for F1 car with spoiler and in this unstructured polyhedral mesh was used and AKN k-ε turbulence model is used. He suggested that 00 spoiler gives less drag than 50.

Shyam P. Kodaliet al. [2] made a numerical simulation of air flow over a passenger car without rear spoiler and compares these results with results obtained for a passenger car fitted with rear spoiler. Pressure based solver the k-ε turbulence model was used in the simulation. The vehicle is operated at various speeds 50,100, 180 and 250 KMPH and then the coefficient of lift is calculated. The vehicle fitted with spoiler gives low coefficient of lift, thereby increase in vehicle stability.

Richard G.J Flayet al. [3] studied the aerodynamic design of a formula SAE race car, the main objective of their research was to increase the down force generated by the vehicle, for that the exhaust was vented into two diffuser tunnels under the car. There by increase in the down force about 30%. They did both numerical modelling as well as the wind tunnel testing.

Luiz Antonio et al. [4] worked on design analysis improvements for a formula SAE racing car. ANSYS CFX software is used to predict the aerodynamic forces like lift, drag and aerodynamic flow path in order to find the way to decrease the drag and improve the down force generated by the car. Wind tunnel test was carried out to validate the results.1:5 reduced scale model was used for wind tunnel testing.

Xu-xia Hu *et al*. [5] carried out a Numerical study on Rear- spoiler of Passenger vehicle. They analysis a passenger vehicle for with and without spoiler. The standard k- ε turbulence model was used to simulate the flow field.24 different cases were simulated and results were compared. The high speed passenger car attached with rear spoiler shows increase in down force as well as reduction in drag.

Sajjad Beigmoradi *et al.* [6] explained the Noise identification for a coupe passenger car by considering spoiler. The main cause for aerodynamic noise is due to fluctuations of pressure on external surface of body. ANSYS FLUENT 14 software is used for simulating the flow. HEXA element type is applied for volumetric meshing. The rear spoiler gives better vehicle stability at high speeds 90,120,140 (KMPH).bumper region produces more aerodynamic noise source and maximum acoustic power level for different relative angles also calculated.

Scott Wordley*et al*. [7] described a numerical, wind tunnel and on-track study for formula SAE car. They increase the down force by designing multi element wing profiles which generates high negative coefficient of lift. Full scale model is used for wind tunnel testing. CFD FLUENT software is used for simulating the flow field.

Mustafa cakir*et al*. [8] made a study on aerodynamic effects of a rear spoiler on passenger vehicle. Sedan car is used for analysis and ANSYS FLUENT software is used for simulating the flow. The spoiler is used for producing more ground effect as well as to reduce the flow separation.

Stefan Bordei*et al*. [9] used various software's like ANSYS FLUENT, CFX, OPEN FOAM is used for simulating flow field and finding out which is less expensive as well as reduces the simulation and gives better results. In accuracy point of view STAR CCM+ gives accurate results for road vehicle.

Sagar Kapadia *al*. [10] worked on detached eddy simulation over AHMED car model. Detached eddy simulation with 25^0 and 35^0 slant angle was chosen for simulation. 35^0slant angle of car was gave a close value of drag as same in the experimental.

The literature review suggested that spoiler's angle of attack will increase the stability and reduce the drag for various drift angles.

II. ANALYTICAL RESULTS AND ANALYSIS OF FORMULA VEHICLE

The numerical analysis of flow over a vehicle is performed commercial software STAR CCM+ solver and the unstructured polyhedral mesh with AKN k-ε turbulence model is chosen for the analysis. le^{-5} is taken as a convergence criteria to get better results. Initially 300 iterations are carried out with first order upwind scheme and relaxation factor for velocity as0.3 and for pressure 0.1 to avoid divergence of the solution. Later the iterations carried out for second order upwind scheme with 0.5 as velocity relaxation factor and 0.3 for pressure till the convergence criteria are met.

III. GEOMETRICAL MODELLING

The vehicle is modelled by using CATIAV5R18 and their different views are shown in Figures from 1-4.

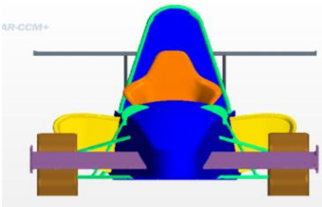
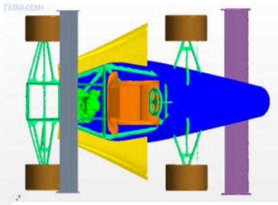
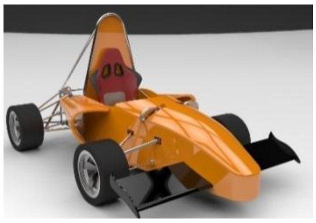
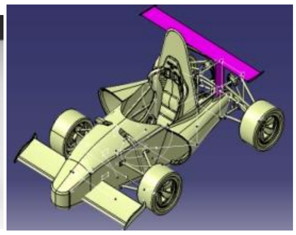

Fig. 1. Front viewFig.2. Top viewFig.3.Isometric view of car with and without spoiler

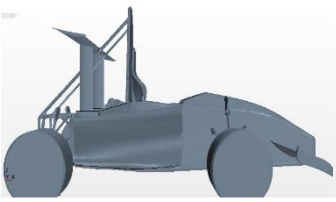
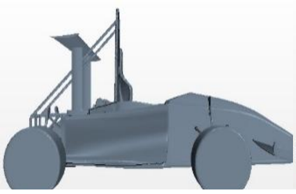
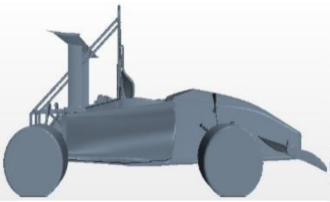
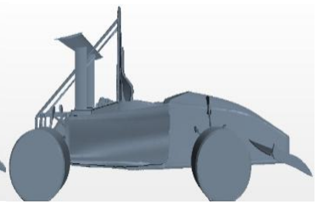

Fig.4 Car with spoiler for -10^0 and 10^0 angle of attackFig.5 Car with spoiler for -5^0 and 5^{0}angle of attack

IV. GRID GENERATION

For the Computational domain Inlet is fixed 3L from the upstream and outlet is specified 5 L from the vehicle in the downstream condition. The top wall is considered as freestream boundary which 4L from the vehicle it is shown in Figure 5.Unstructured polyhedral mesh is used for the numerical analysis. To resolve the boundary and reducing the computational time more grid made near the car and spoiler for betterment of results and capture the vorticity near the vehicle and the spoiler is shown in Figure 6 and mesh on vehicle is shown in Figure 7.

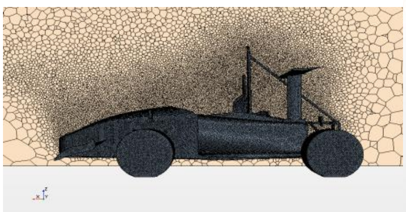
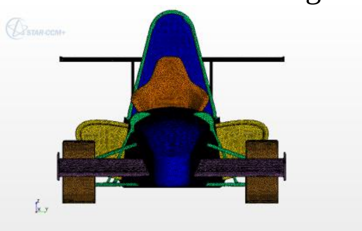

Fig. 6. Computational domainFig.N 7 Sectional view of meshFig. 8 Front view

V. BOUNDARYCONDITION

The analysis of vehicle was carried out with the following boundary condition by using STARCCM+ CFD software. In that analysis only straight wind condition was considered at three different speeds 75,100,125 KMPH. Constant velocity, zero gauge pressure were applied as a inlet and outlet conditions. The boundry condtions used in that analysis is and the computational domain is shown Table 1.

Table 1 Boundary conditions

BOUNDARY	BOUNDARY CONDITIONS	VALUES
Inlet	Constant velocity	V=75,100,125 KMPH
Outlet	Pressure outlet	Const. pressure=0 N/m2
Car body	No slip – stationary wall	-
Domain	Constant velocity	V=75,100,125 KMPH

VI . RESULTS AND DISCUSSIONS

Grid independent test is carried out for the formula vehicle with 1.6 million, 2.2million and 2.7 million cells. AKN k-ε two layer models were used for the test. The drag coefficient values along the surface are measured. From table it can be seen that the values at 1.6 million is deviating from the values obtained with 2.2million cells. However, difference between the values obtained from 2.2million cells with those 1.6 million cells is very less (less than 5%) [1]. Hence, 1.6 million cells is considered for further analysis for formula vehicle. The vehicle is analyzed with different speed as well as with different drift angles and various spoiler angle of atttack. Drag force is measured for various drift angle is shown.

From the contour diagram for -5^0 spoiler angle of attack with 0^0 drift angles, the variation of the pressure and velocity of flow over the car is seen. At the frontal area of the car, 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 areas due to curvilinear nature of the vehicle is shown in Figures from 7 &8.

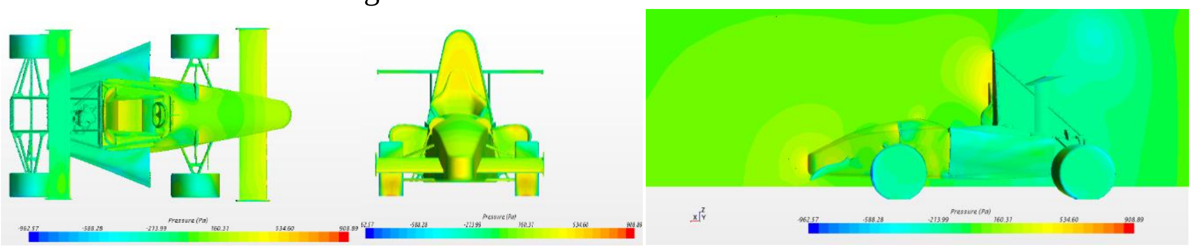

Figure 7 pressure Contours on the surface of the car

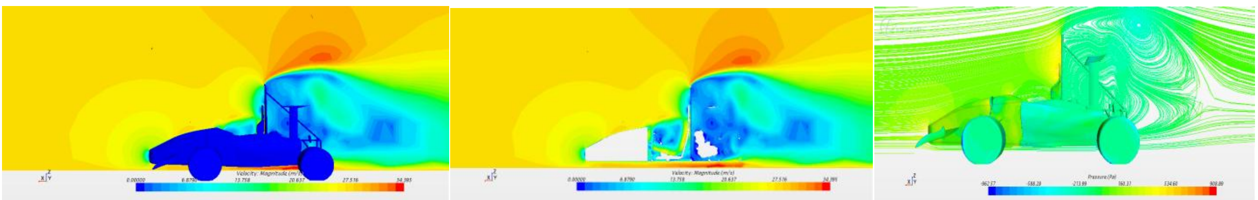

Figure 8 Velocity Contours on the on the surface of the car and section view of computation domain

VII. CONCLUSION

In this work, the formula car is analysed without spoiler as well as with spoiler for three different angles form -10^0 to 10^0. In this analysis, polyhedral mesh with AKNk-ε turbulence model is used. The drag and lift forces areobtained from this analysis for with and without spoiler. It is clearly seen that -5^0 deflected flap with 10 KMPH gives less drag and more amount of ground force (negative lift) for stabilization than other angle of attacks.

REFERENCES

[1] Numerical simulation of drag reduction in formula 1 car by A.Muthuvel, N.Prakash, GodwinJohn,International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 ,National Conference on Advances in Engineering and Technology(AET- 29th March 2014)
[2] Numerical simulation of air flow over a passenger car and the Influence of rear spoiler using CFD Shyam P. Kodali and Srinivas Bezavada International Journal of Advanced Transport Phenomena Vol. 01, No. 01, Jan-Dec 2012
[3] .Aerodynamic design of a formula SAE race car by Richard G.J Flay and Andrew Hammond, BBAA VI International Colloquium on: Bluff Bodies Aerodynamics & Applications Milano, Italy, July, pp. 20-24 2008.
[4] Design Analysis improvements applied to a formula SAE racing car Aerodynamicsby Luiz Antonio Negro Martin Lopez 2011.
[5] A Numerical Study On Rear-spoiler Of Passenger Vehicle Xu-xia Hu, Eric T.T. Wong World Academy of Science, Engineering and Technology Vol: 57 2011-09-25.
[6] Aerodynamic Noise Source Identification for a Coupe Passenger Car by Numerical Method Focusing on the Effect of the Rear Spoiler by SajjadBeigmoradi, KambizJahani, ArashKeshavarz and Mohsen BayaniKhaknejad CAE Engineer, R&D Center of SAIPA in SAE International 2013..
[7] Aerodynamics for formula SAE: A Numerical, wind tunnel and on- track Study by Scott Wordley and Jeff Saunders. SAE paper 2006-01-0808
[8] CFD study on aerodynamic effects of a rear wing/spoiler on a passenger vehicle by Mustafa cakir (2012).Santa ClaraUniversity. Department of automobile engineering MS thesis.
[9] Aerodynamic results for a notchback race car by ȘtefanBordei, Florin PopescuThe Annals of "Dunărea De Jos" University of Galați Fascicle V, Technologies in Machine Building, ISSN 1221- 4566, 2011.
[10] Detached-eddy simulation over a referenceAhmed car model by Sagar Kapadia and Subrata Roy, Matthew Vallero, KennethWurtzler and James Forsy.

