Proceedings of the World Congress on Engineering 2008 Vol II
WCE 2008, July 2 - 4, 2008, London, U.K.

CFD Investigation of Airflow on a Model Radio Control Race Car

Chainani. A, Perera. N

***Abstract*— The modern day design of vehicles, especially in the racing industry involve a great deal of air flow study. This study shows that drag force adversely affects the forward motion of the car and that there is a difference in the pressure between the air flowing above and below the car. This produces forces along the vertical axis. Aerodynamic forces acting on a car greatly reduces its efficiency. If the car is redesigned to optimise these forces it could produce better results. This paper discusses various techniques that have been used to redesign and optimise the aerodynamics of a model radio control race car.**

***Index Terms*— Computational fluid dynamics, down force, drag force, (CFD), vehicle aerodynamics**

I. INTRODUCTION

Until the mid 1960's [1] the understanding of vehicle aerodynamics was very limited and the shape of the car was left in the hands of the designer. The cars were "boxy" and had large frontal areas. The growth of computing technology in the 1970's and 80's [1] allowed car manufacturers to analyse air flow around cars and recognize the importance of aerodynamics in vehicle development. This progress allowed further research to be focused on reducing the drag coefficient. It has been shown [2] that 40% of the drag coefficient is dependant on the external shape of the car and is concentrated at the rear of the geometry.

It has been established [3] that a systematic aerodynamic study of the various parts of a vehicle can help improve its aerodynamics. Also, computational techniques of estimating airflow aided with comprehensive experimentation can produce sustainable results. Researchers [2] have also investigated the effects of utilising various geometries for the components of the airfoils behind cars. Others [4] have investigated the wing/body interaction in generic shapes of closed-wheel race-cars. There have also been investigations in the patterns of airflow in case of convertible cars [5]. However, there still is great potential for research in this application of fluid dynamics.

A moving car experiences an increase in aerodynamic forces with an increase in its velocity. Just like an airfoil, the body of a car experiences drag and lift forces, the only major difference being that due to the shape of a car it experiences a negative lift or down force [6]. The down force generated by its shape gives the car the ability to go around non-banked curves with higher speeds. Hence an aerodynamically superior car design generates a low drag force but a high down force. An aerodynamically superior car design can reduce fuel consumption and increase its efficiency. This produces a more environmentally friendly car. However due to the competitiveness of the motor sport industry, there is limited current research findings pertaining to the aerodynamic features of race cars.

The subject of this paper is to investigate and analyse the airflow around a radio control model race car using a commercial computational fluid dynamics (CFD) software package.

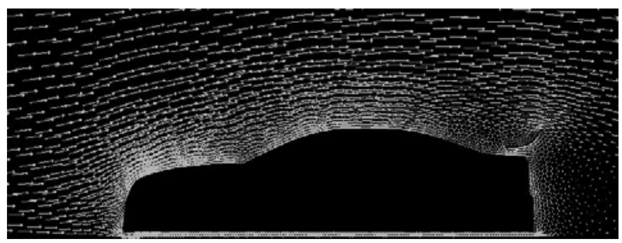

Fig.1 Airflow over the model race car

II. NOMENCLATURE

V_{in} = Inlet Velocity

P_{out} = Outlet Pressure

ρ = Density

F_d = Drag Force

F_l = Lift Force

A = Area

C_d = Drag Coefficient

C_l = Lift Coefficient

C_s = Lift Coefficient

ε = Epsilon

III. THEORY

The basis of aerodynamics is the Bernoulli's equation which states that at any point in a streamlined flow if the local air stream is lower than that of the undisturbed flow, the loss is compensated by an increase in pressure [1].

The authors are with School of Computing, Engineering and Information Sciences, Northumbria University (Email: amit.chainani@unn.ac.uk, noel.perera@unn.ac.uk)

As the body of the model car is not streamlined, it causes local disturbances in the air flowing around the body. The fluctuation in the local air stream velocity produces a spectrum of pressure on the various faces of the vehicle. This pressure creates a rise in forces acting along the three axis [1]. The drag force is given by:

$F_d = (\rho V^2 A C_d) / 2$

The relative velocity of the air mass to the car is the same as the relative velocity of the ground to the car. Hence it is assumed that the velocity of the air mass is equal to the relative velocity of the ground to the car.

In this investigation the C_d value is assumed to be a constant since the general airflow is turbulent.

Similarly, for the Lift force is given by

$F_l = (\rho V^2 A C_l) / 2$

And Force caused by cross winds, side force is given by

$F_s = (\rho V^2 A C_s) / 2$

As shown [6] the flow of air in the boundary layer of a car has been found to be turbulent for the most part. The boundary layer causes particle stagnation around some regions along the body of the car. This effect is known as "skin friction" [6]. Skin friction is attributed to the viscous forces which significantly alter the flow of air around the car. Hence it is essential to consider viscous forces and model the viscosity of air while analysing the air flow around the car. Therefore to model viscosity, the Navier-Stokes equations were incorporated into the CFD analysis.

IV. Methodology

A velocity test was carried out in the laboratory to obtain the average velocity of the model car. The velocity test ensured that the boundary condition (V_{in}) used in the CFD model replicated the actual velocity of the model race car.

A commercial CFD software package was used to analyse the airflow around the model race car. The body of the car was scanned using a 3-D scanner to with a precision of 1 mm i.e. 0.233%. Due to scanner limitations, the rear spoiler of the car was removed during the scanning process. The rear spoiler was generated using Solid Works a 3D mechanical CAD software package. The scanned data was imported to Solid Works and checked for errors and self intersecting geometries. The spoiler was then added to the scanned model and the entire geometry was exported to the CFD package. The CFD model did not incorporate the airflow around the wheels and assumed the body of the car to be a shell without wheel arches. This model also assumes no side, up or down winds.

A redesigned car body was used to further analyse and compare the aerodynamic effects of a change in the body shape. This redesigned car body has the same vital dimensions i.e. length, width, average height, ground clearance and volume of the model car.

V. The Experiment

The model car was tested for the average velocity which was obtained over 15 straight runs for a distance of 9.144 m (30 ft.). The velocity test was carried out in a laboratory under controlled conditions. Due to the experiment being carried out indoors the cross wind effect was omitted. The table of results is shown in Table 1.

Serial No.	Time (s)	Distance (m)	Velocity (m/s)
1	2.72	9.144	3.36
2	2.84	9.144	3.22
3	3.34	9.144	2.74
4	2.54	9.144	3.60
5	1.72	9.144	5.32
6	2.47	9.144	3.70
7	2.06	9.144	4.44
8	2.06	9.144	4.44
9	2.13	9.144	4.29
10	2.35	9.144	3.89
11	2.22	9.144	4.12
12	2.06	9.144	4.44
13	2.29	9.144	3.99
14	1.78	9.144	5.14
15	1.75	9.144	5.23

Table 1. Results of the velocity test

VI. CFD Analysis

The results obtained from the experiment were used as boundary conditions for the CFD model. A symmetry section of the model race car as shown below in Fig. 2 was initialised using the boundary conditions V_{in} and P_o.

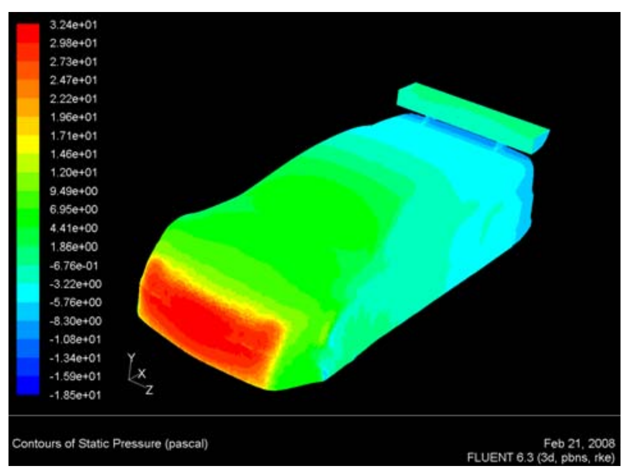

Fig. 3 Contours of static pressure for the model race car.

As demonstrated in Fig.3 the front end of the car experiences significantly higher pressures than the rest of the car. This can be attributed to its poor curvature and high projected area. It is also seen that the pressure head on the

ISBN:978-988-17012-3-7

rear airfoil of the car does not generate the down force that is required from an airfoil. Therefore the inclusion of the rear airfoil apart from its aesthetic value has no significant aerodynamic contribution to the model race car. It is also shown that there is a higher pressure concentration on the front end of the car that contributes to an increase in the overall C_d value.

Stylish and protruding features such as overhanging curves along the side of the car as indicated in Fig. 4 produce unwanted regions of high pressure.

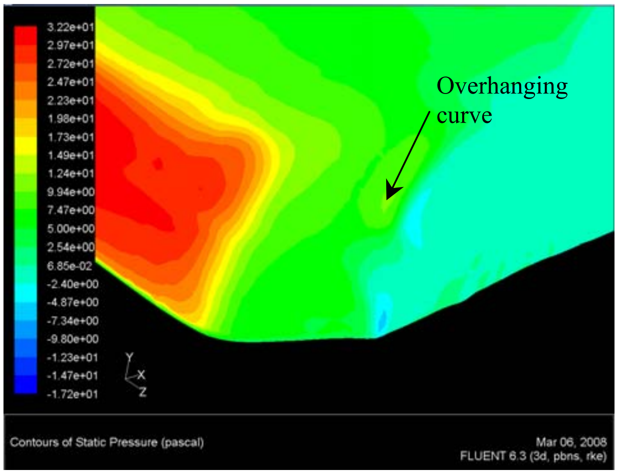

Fig. 4 Static pressure contour of the overhanging curve

The Fig.5 demonstrates that the velocity of the air reduces as it reaches the front of the car. This velocity then increases due to a positive slope above the front end of the car. The airflow velocity over the roof reduces significantly and remains constant as it flows towards the rear of the car. This airflow trend is attributed in part to the rear airfoil which does not trip the flow or decrease its velocity.

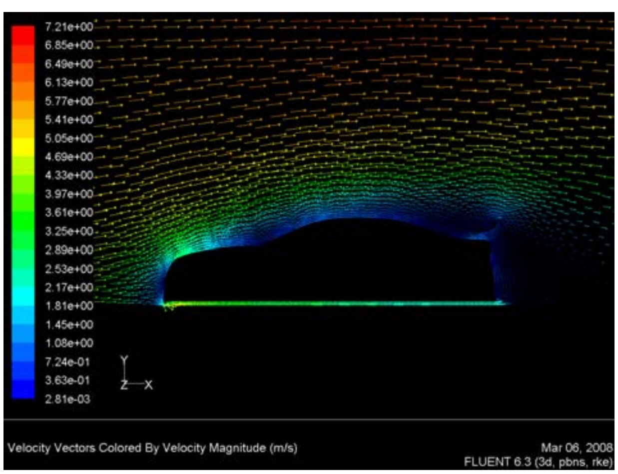

Fig. 5. Velocity vectors of the airflow over the model race car.

The CFD analysis on the model race car has shown that the rear airfoil, overhanging arches and frontal shape of this model race car has contributed to its ineffective aerodynamic performance.

The contours of static pressure shown in Fig. 6 demonstrate that the redesigned model race car has a reduced pressure head in the front end. This pressure head greatly contributes to the down force generated in the front half of the car therefore provides for stability. As the air flows beyond the front grille of the car, the static pressure reduces with an increase in the gradient of the bonnet. This graduated change in the pressure contours reduces the concentrated pressure heads along the upper surface of the redesigned model race car. The reshaped

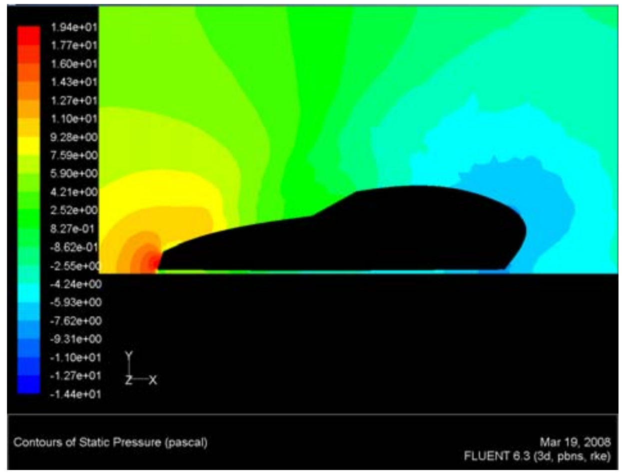

Fig. 6 Contours of static pressure for the redesigned model race car

front grille on the redesigned model race car acts like a flow tripper [7] delaying the formation of a potentially stagnant boundary layer. This is demonstrated when comparing Fig. 5 with Fig. 7. This reduces the viscous forces caused by the boundary layer of the car therefore contributing to a reduction in the overall C_d value.

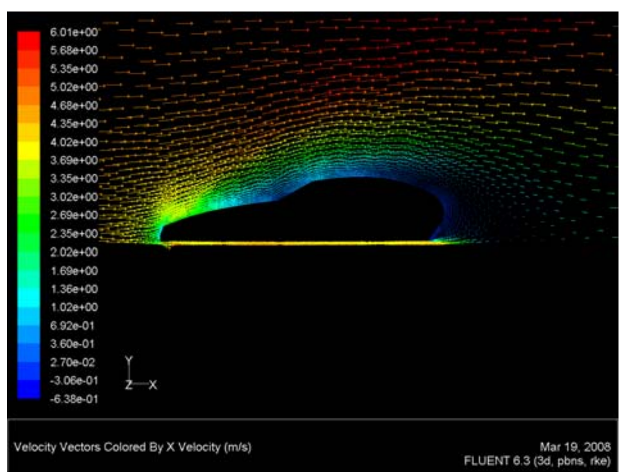

Fig. 7 Velocity vectors of the airflow over the redesigned model race car

Another essential change made along the side of the car is the elimination of protruded side curves that acted as surfaces of high static pressure as shown in Fig.4. The redesigned car has side surfaces that are streamlined i.e. shaped like a fish which allow for a smoother transition of air.

The reshaping of the rear end of the car has been greatly influenced by a designing technique called "bob-tailing" [6]. As shown in Fig. 5 and Fig. 7 the wake i.e. region of air recirculation is at the back of the car. In this region both models experience high turbulence and reverse velocities. However "bob tailing" the back reduces the boundary layer stagnation and greatly changes the flow pattern behind the car. This causes a reduction in the wake region which should contribute to a decrease in the drag force.

VII. Conclusion

This investigation has demonstrated that there is a significant change in the coefficients of lift and drag of the model race car when a more streamlined body design is adopted. The results obtained show a 40.49% reduction in drag coefficient with a loss of 24% in the down force. This further verifies previous research [9-11] indicating that a body having an "inverted tear-drop" shape generates a lower drag force than one with a "conventional" shape. The findings of this investigation also confirm previous research [12-15] detailing that bob-tailing' the rear end of the car reduces the recirculation region behind the car. This further reduces the drag force acting on the car. Furthermore it was also determined that varying the angle of the front grille has an affect on the down force acting on the car.

VIII. Further Work

Suggestions for future work are:

- Verifying the computational results by building and testing both models in a wind tunnel.
- Building a practical model of the redesigned race car and carrying out experimental tests on it to verify the obtained computational results.
- Gathering data on the actual environmental conditions that the car is exposed to and using that data in the CFD analysis.

References

[1] Wolf-Heinrich Hucho, "Aerodynamics of Road Vehicles". Cambridge: Butterworths, 1987, ch. 1-4.
[2] W. Kieffer, S. Moujaes, N. Armbya "CFD study of section characteristics of Formula Mazda race car wings" March 2005
[3] F. Muyl, Laurent Dumas, Vincent Herbert "Hybrid method for aerodynamic shape optimization", June 2003
[4] G.de Vahl Davis and C. Fletcher, "Computational Fluid Dynamics". Amsterdam : Elsevier Science Publishers B.V., 1988, pp 315 – 328
[5] Hiroshi China, Masahiro Yoshida, Morihiro Takada, Kunio Nakagawa, "Air flow control around the cabin of a convertible car" July 1994
[6] AJ Scibor - Rylski, "Road Vehicle Aeerodynamics" London: Pentech Ressl, 1975, ch 1-7.
[7] Masaru Koike, Tsunehisa Nagayoshi, Naoki hamamoto,"Research on Aerodynamic Drag Reduction by Vortex Generators"
[8] Appupillai Baskaran and Ahmed Kashef, "Investigation of air flow around buildings using computational fluid dynamics techniques", 1996
[9] Heller, A. "Der neue Kraftwagen von Dr. Ing Rumpler", 1921
[10] Eppinger, A. "Tropfenwagen- Anwendung der Flugzueg Aerodynamik", 1921
[11] Rumpler, E. "Das Auto im Luftstrom", 1924
[12] Stapleford, W. R. and Carr. "Aerodynamic characteristics of exposed rotating wheels" 1970
[13] W. Kieffer, S. Moujaes, N. Armbya "CFD study of section characteristics of Formula Mazda race car wings" March 2005
[14] Morelli, A. " Aerodynamic actions on an automobile" 1969
[15] Loghton, C.J. "Rotating wheel experiment", Unpublished, City University, London, 1969