CFD Analysis and Optimization of a Car Spoiler

C.V.Karthick Bala Murugan[1], P.A.Nigal Ashik[2], P.Raju[3]

Assistant Professor

Abstract

Aerodynamic characteristics of a race car is of significant interest in creating negative lift for stability and in reducing drag for achieving high speed . This paper presents a numerical simulation of flow around racing car with spoiler positioned at the rear-end using commercial fluid dynamics software FLUENT 6.3.26. The study focuses on Computational Fluid Dynamics (CFD) based lift and drag prediction on the car body and on improvement in the design due to spoiler configurations. A three dimensional computer model of a racing car was used as the base model in this study. From the results obtained, it was found that at the certain height of spoiler and wind collision angle, the change in C_D is negligible but C_L changes significantly as the speed of the racing car increases. Furthermore, the values of C_D and C_L increase as the height of the spoiler decreases and the angle of wind collision increases.

Keywords: aerodynamics, numerical simulation, CFD.

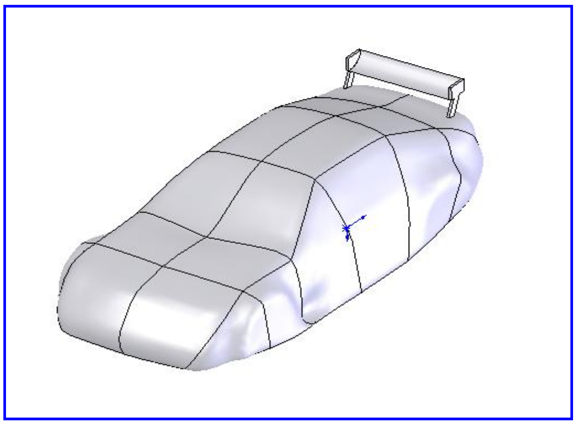
1. Introduction

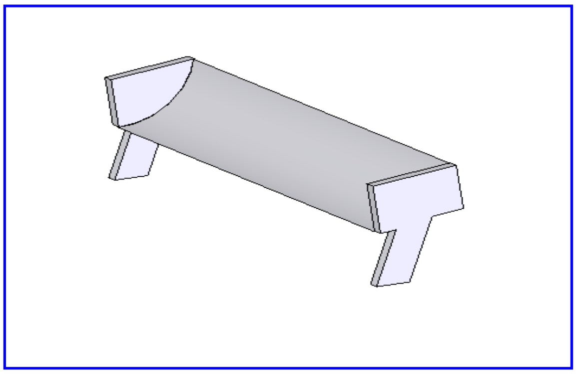
At present, modified car racing becomes more popular around the world. Aerodynamic characteristics of modified racing car is therefore inevitably of significant interest. Well design of racing car obviously provides a reduction in car racing accident and fuel consumption. Because experimental study on aerodynamic of vehicle is cost effective, considerable efforts have been invested to study vehicle aerodynamics computationally. Basara [1] presented calculations of the flow around different shapes of vehicles. In his study, different meshes, differencing schemes and turbulence models were used to avoid possible misleading in prediction analysis. Basara [1] showed that the results were encouraging for the Reynolds-Stress model as importance of having anisotropic formulation of turbulence. The quality of k-ε results is certainly very case dependent. It can only accidentally produce good results which may happen when the flow does not separate or slightly separates from the surfaces and where vertices are present on such positions that have no strong influence on the pressure distribution on the car .This paper presents a numerical analysis of flow around modified racing car with spoiler positioned at the rearend using commercial fluid dynamics software FLUENT 6.3.26. The study focuses on CFD-based lift and drag prediction on the car body and an improvement in the design due to spoiler configurations.

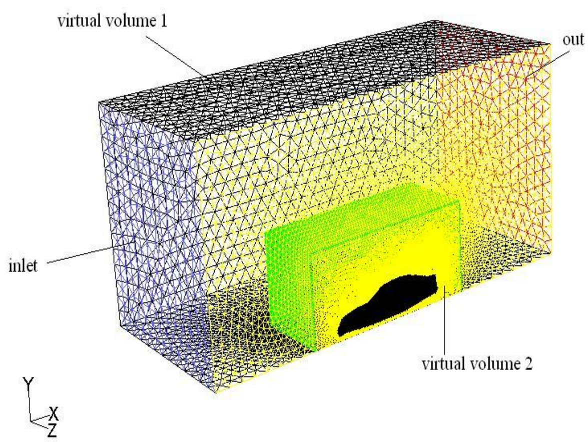
2. CFD Model Setup

Figure 1 and Figure 2 show 3-D computer models of racing car and spoiler respectively. Virtual volume 1 was created using this racing car model, cut along its symmetry plane and inside a small rectangular box. Geometry of the wind tunnel was added around virtual volume 1 and named virtual volume 2. Both virtual volume 1 and volume 2 were created in SolidWorks and have the same symmetry plane with the racing car model as shown in Figure 3. A fine tetrahedral mesh was created in Gambit around virtual volume 1. This fine mesh is necessary to be able to capture aerodynamic effects around this important region. A size function in Gambit was applied to virtual volume 2 for adding more cells nearer to the racing car and lesser cells towards the less important areas at the outer edges of the wind tunnel. Figure 4 shows the resulting mesh looking in three dimensions.

Boundary conditions for the faces and volumes were configured in Gambit. The velocity of the air at the inlet was set in the range of 80 km/hr to 200 km/hr. The outlet boundary was set as pressure outlet 0 Pa. Surface of the racing car was set as wall and the cut half surface was set as symmetry as shown in Figure 4. The density and viscosity of air are 1.225 kg/ m^3 and 1.7894 × 10^{-5} kg/ms respectively. As suggested by Basara 1999, the Reynolds-Stress model was used to capture turbulent flow. Residual conditions were all set to 10^{-3} for convergence criteria. Spoiler was positioned at the rearend at 4 different heights and was adjusted to wind collision angles of -10 to -2 degrees at 2 degree increments.







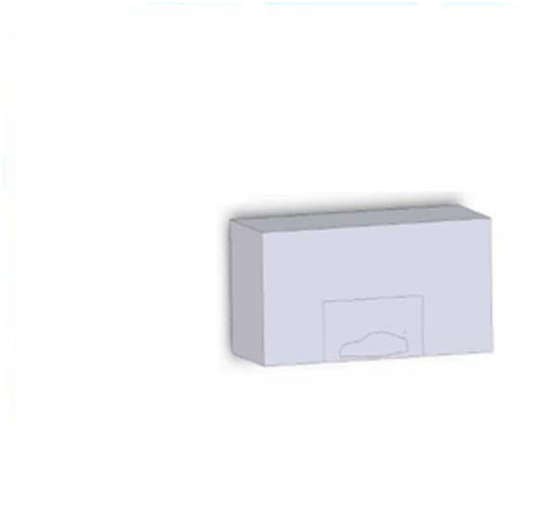


Figure 1. Original SolidWorks model of racing car
Figure 2. Original SolidWorks model of racing car spoiler.
Figure 3. Boundary condition of racing car
Figure 4. Resulting mesh looking in three dimensions.

3. Results

3.1 Effects of racing car speed on drag coefficient (C_D) and lift coefficient (C_L).

Figure 5 shows the variations of C_D over the racing car speed ranges from 80 km/hr to 200 km/hr at the wind collision angle of 0 degree and the spoiler height of h/H = 0.5 and h/H = 1 where h is the height from the rearend to the spoiler and H is the height from the rearend to the roof of the racing car. It was found from this figure that the change in C_D is negligible as the speed of the racing car increases no matter what the spoiler height is.

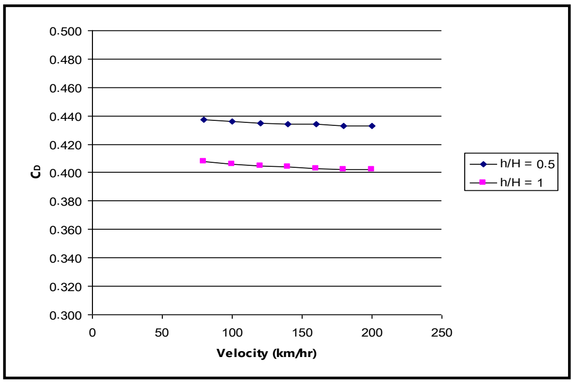


Figure 5. Variation of C_D over speed range of racing car.

However the downward force acting on the racing car with the spoiler at the rearend increases significantly as the speed of the racing car increases. This fact also applies to the case of different spoiler height at the same wind collision angle as shown in Figure 6.

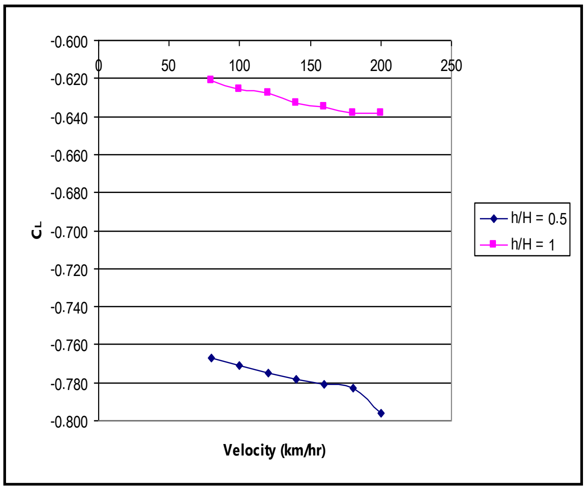


Figure 6 .Variation of C_L over speed range of racing car.

3.2 Effects of spoiler height on C_D and C_L.

It is clear from Figure 7 that C_D decreases as the spoiler height increases for any value of racing car speed. Figure 8 and Figure 9 illustrate the velocity distribution over the racing car for the spoiler height of h/H = 0.25 and h/H = 1 respectively. From both figures, it was found that there are recirculation zones behind the rear-end of the racing car. By comparing Figure 8 and Figure 9, the recirculation zone behind the rear-end of racing car with spoiler height h/H = 0.25 is clearly larger. Hence the pressure behind the rear-end of the racing car is also lower as shown in Figure 11. This contributes to greater pressure difference between the front and rear-end of the racing car. Therefore at the same speed and wind collision angle, the racing car with lower spoiler height experiences higher drag or C_D.

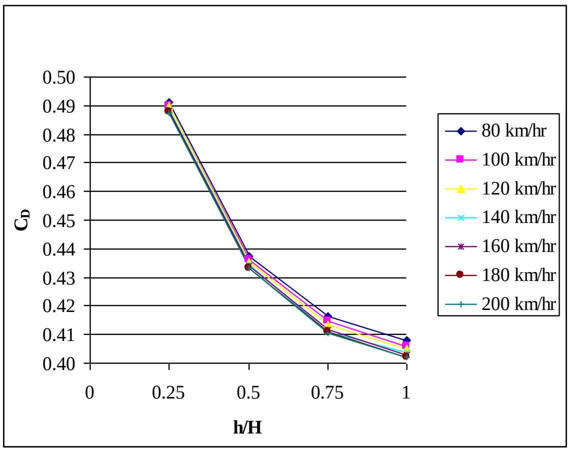
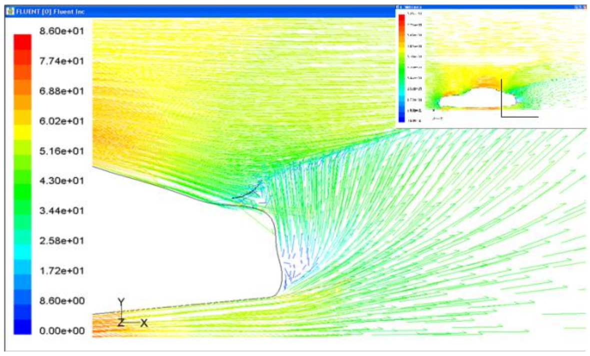


Figure 7. Variation of C_D over the range of spoiler height



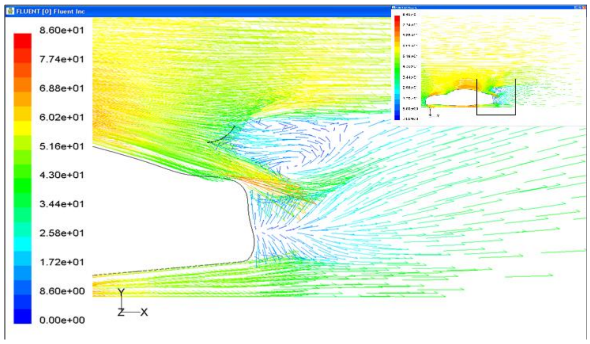


Figure 8 Velocity distributions over racing car moving at 200 km/hr with spoiler positioned at h/H = 0.25

Figure 9 Velocity distribution over racing car Moving at 200 km/hr with spoiler positioned at h/H= 1

It should be noted from Figure 10 that C_L decreases as the height of the spoiler increases independently of racing car speed. Figure 11 and Figure 12 show the pressure distribution over the racing car moving at the speed of 200 km/hr with the spoiler height of h/H = 0.25 and h/H = 1 respectively. It was shown in Figure 11 that the pressure over the rearend of the racing car is smaller compared to those plotted in Figure 12. This contributes to lower downforce over the racing car at the same speed and wind collision angle.

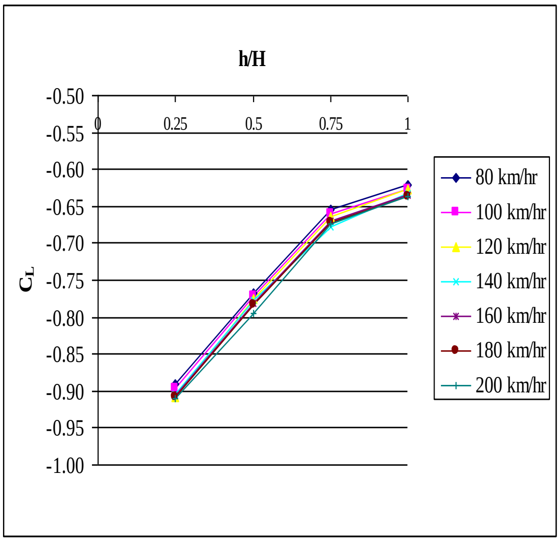
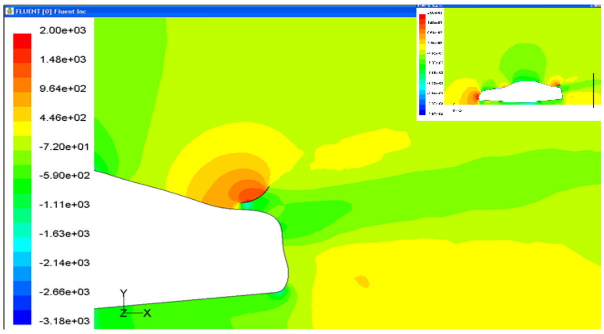


Figure 10 Variation of C_L over the range of spoiler height



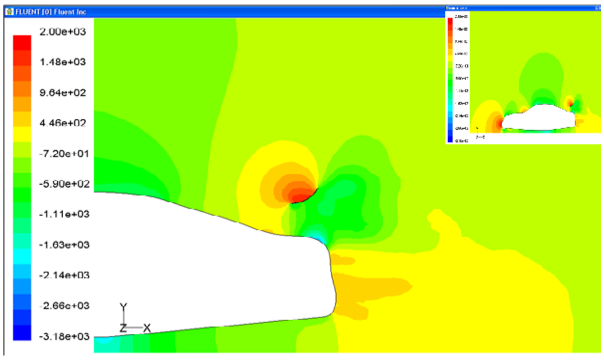


Figure 11 Contours of pressure over racing car Moving at 200 km/hr with spoiler positioned at h/H = 0.25

Figure 12 Contours of pressure over racing carmoving at 200 km/hr with spoiler positioned at h/H= 1

3.3 Effects of wind collision angles on C_D and C_L.

It is clear from Figure 13 that C_D increases as the angle of wind collision increases. By comparing the velocity distributions plotted in Figure 14 and Figure 15, it was found that at a particular spoiler height the spoiler that posses smaller angle of wind collision gives higher drag and C_D. This is due to the fact that with smaller angle of wind collision, the spoiler would creates smaller recirculation zone behind the rearend of the racing car. This implies to higher pressure behind spoiler but lower pressure behind the rearend of the racing car.

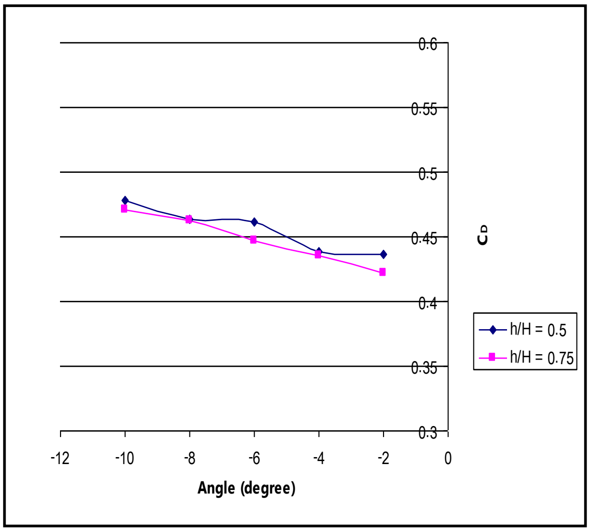
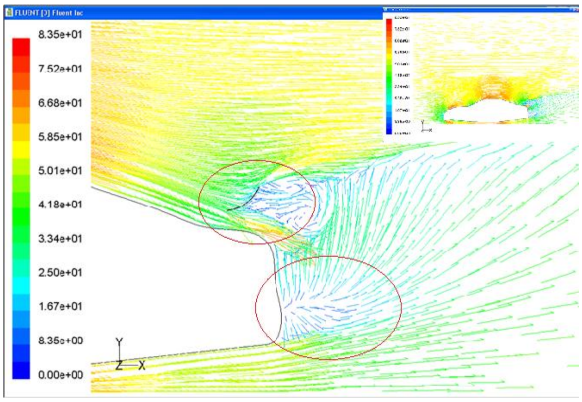


Figure 13 Variation of C_D over the range of wind collision angle



International Journal of Mechanical Engineering and Research, ISSN 0973-4562 Vol. 5 No.1 (2015)
© Research India Publications; http://www.ripublication.com/ijmer.htm

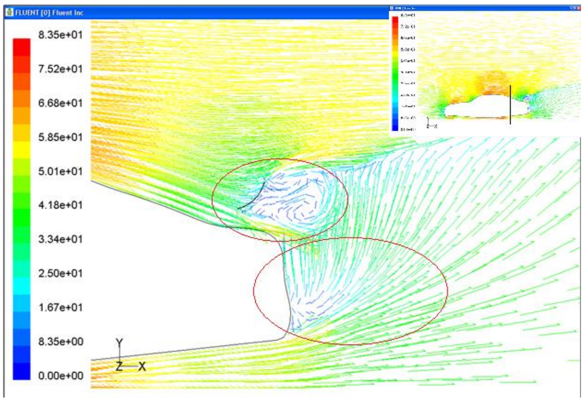


Figure 14. Velocity distribution over racing car moving at 200 km/hr with spoiler positioned at -2 degree of wind collision angle and h/H = 0.25

Figure 15. Velocity distribution over racing car moving at 200 km/hr with spoiler positioned at -10 degree of wind collision angle and h/H = 0.25

Moreover Figure 16 shows that C_L increases as the angle of wind collision increases. Also from the contours plot of pressure as shown in Figure 17 and Figure 18, it is clear that the pressure distribution over the rearend of the racing car is higher for the wind collision angle of –2 degree than that of -10 degree. This therefore contributes to higher downforce to the racing car

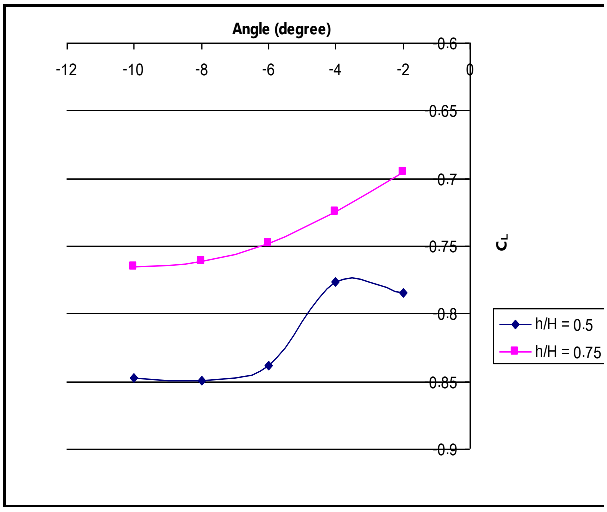
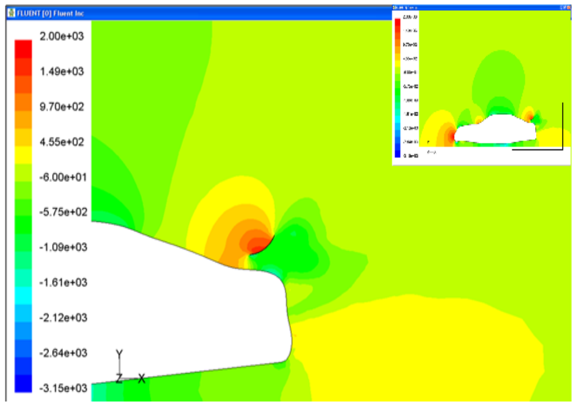


Figure 16 Variation of C_L over the range of wind collision angle



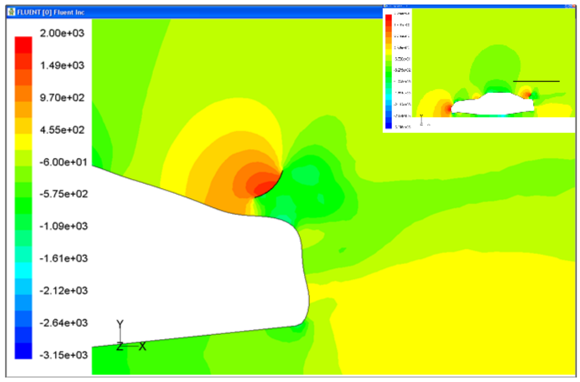


Figure 17. Contours of pressure over racing car moving at 200 km/hr with spoiler positioned at -2 degree of wind collision angle and h/H = 0.5

Figure 18. Contours of pressure over racing car moving at 200 km/hr with spoiler positioned at -10 degree of wind collision angle and h/H= 0.5

4. Conclusion

Computational Fluid dynamics simulation using FLUENT to predict flow around modified racing car has been achieved. It is clear from the results obtained that at a certain height of spoiler and wind collision angle, the change in C_D is negligible as the speed of the racing car increases. However the downward force acting on the racing car with the spoiler at the rearend increases significantly lower as the speed of the racing car increases. Moreover the lower spoiler height tends to gives both higher C_D and C_L. Finally it was found that at a particular racing speed and spoiler height, C_D and C_L increase as the angle of wind collision increases.

References

[1] B. Basara, Numerical Simulation of Turbulent Wakes around a Vehicle, "ASME Fluids Engineering Division Summer Meeting", California, USA, 1999

[2] John D Anderson Jr", Fundamentals of Aerodynamics", Tata McGraw hill publishing company Pvt.ltd, New Delhi, 2007.

[3] John D Anderson Jr," Computational fluid dynamics: The basics with applications", Tata McGraw hill publishing company Pvt.ltd, New York, 2004.

[4] Hucho WH."Aerodynamics of Road vehicles" SAE international, 1998.

[5]"Fluent 6.3 user guide", Fluent Inc.

[6] Joseph Katz," Aerodynamics of racecars", *Annual Rev. Fluid mech*., 2006, 38:27-63.

[7] www.f1complete.com