NAME SUDEEP KUMAR
REG NUMBER 16BME1254

Aerodynamic Drag Reduction Study of Formula Race Car

Abstract

Aerodynamic drag is one of the major problem to accelerate a solid body when it runs in the air. In this context we talk about various drag reduction techniques which can be applied in a formula race car. When a racing car burns fuel to accelerate, drag force pulls it to reduce the speed and hence the fuel efficiency is adversely affected. About 50 to 60% of total fuel energy is lost only to overcome this adverse aerodynamic force. The drag force can be reduced by optimizing its shape to ensure stream-lining. Reduction of aerodynamic drag has become one of the prime concerns in vehicle aerodynamics This article is concentrated on different aspects analysis of aerodynamic drag of racing cars and different drag reduction techniques such as rear under body modification and exhaust gas redirection towards the rear separation zones

Through a numerical process (Finite Volume Method) of solving the Favre-averaged Navier-Stokes equations backed by k–epsilon turbulence model, the drag coefficient of the car under analysis is found to be 0.3233 and it is evident that the drag can be reduced up to 22.13% by different rear under-body modifications and up to 9.5% by exhaust gas redirection towards the separated region at the rear of the car. It is also evident that if somehow the negative pressure area and its intensity at the rear of the car can be minimized, the separation pressure drag is subsequently reduced.

The main goal of this study is to enhance the stability of the vehicle and reduce the drag. The CFD analysis is done on full scale model. The aerodynamic study is conducted in the ANSYS Fluent software to perform a turbulent stimulation (using k - ϵ model) of the air flow on the SAE car. The results are graphically shown with co – efficient of drag, velocity contour.

Keywords: Drag reduction; CFD analysis; SAE car modification: Aerodynamic drag; separation; coefficient of drag; under-body diffuser; exhausts gas redirection

1. Introduction

Aerodynamic drag of racing cars has probably become the area of interest over last five decades in experimental and practical field of fluid dynamics. In order to get a good performance, the vehicle has to aerodynamically efficient and hence many researchers and authors have described different forms of the drag, possible reasons behind them and several ways of minimizing the drag. The drag force is the unwanted thing which normally acts against the drive force of the car. The down force is useful to maintain the race car in ground. Some numerical simulations are performed to analyse the drag of a racing car and some procedures were followed to reduce it like reducing the flow separation.

The capability of the aerodynamic engineer is to compromise between down force and drag. Initially the SAE car is analyzed and co – efficient of drag is found. In the second case the seats are slotted to reduce the drag. In the third case the spoiler is fitted to increase the stability of the vehicle.

Aerodynamic drag (C_d):

When the fluid flows over the surface, the surface will resist its motion. This is called drag. Aerodynamic drag is the sum of pressure drag and viscous drag. The pressure drag is the most dominant one of the both. The pressure drag is causes due the shear forces acting between the two layers of fluid.

$$\text{Aerodynamic drag } C_d = \text{Drag force}/0.5\rho v^2 A$$

Where ρ is the air density in kg/m^3, A is the effective frontal area in m^2, v is the velocity in m/s. The CFD is the effective tool which reduces times and is increasing importance. There are two different types of flow, laminar and turbulent. Laminar flow is smooth where the adjacent layers of fluid will undergo sliding between each other. The turbulent is the chaotic and random. It is normally unsteady, dissipative and 3 dimensional. Reynold's number decides whether the flow is laminar or turbulent. Over the surface there is a thin layer of air exists. The velocity of the air is

reduces due the surface. This layer above the surface is called boundary layer. The length of the boundary layer increases as we move away from the surface. The boundary layer initially will be laminar and transforms into turbulent as it moves further over the larger distance. When there are sudden change in the surface this would lead to flow separation. Flow separation normally takes place when the upper layer of the fluid can no longer pull the lower layer of fluid within the boundary layer.

The flow can be described by Navier-Stokes equation:

$$\rho\left(\frac{\partial u_i}{\partial t}+u_j\frac{\partial u_i}{\partial x_j}\right)=-\frac{\partial p}{\partial x_i}+\mu\frac{\partial^2 u_i}{\partial x_j\partial x_j}+f_i$$

Where u is the velocity, t is the time, x is the position, p is the pressure and ρ is density in kg/m3 [2]. The same thing happens in the case of side view mirror. The flow impacts on the back side of the mirror, the flow get divided. As the flow moves further the flow separation takes place and large amount wake will be formed at the front part of the mirror. With the reduced scale the characteristic length gets reduced, hence the Reynold's no. also reduces. Further there should be geometrical similarity for the reduced scale model. Computational Fluid Dynamics is initial step before the wind tunnel validation is done. There are different methods to solve the flow. The most common of those are –

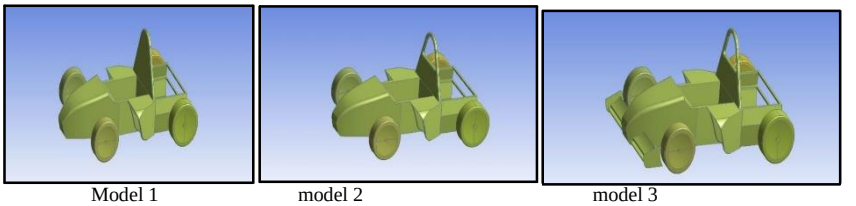
- Direct Numerical Simulation (DNS) – It solves the Navier-stokes equation numerically. It requires very fine grid size and requires very high computer memory. It can solve all turbulence scales.

- Reynold's Averaged Navier-Stokes Equation (RANS) – It gives the approximate time averaged solution to the Navier-Stokes equation. The velocity field is fluctuating and it cannot solve all turbulence scales.

- Large – eddy Simulation (LES): It solves the large eddies. The simulation is time dependent.

Design Modifications

The first solid model is the basic SAE car model outlining the overall shape with actual dimensions;
The second solid model is the SAE car model with cut out in the firewall to reduce drag. The third solid model is the SAE car model with cut out in the firewall and a front wing. This will reduce drag and increase stability by increasing the down force.

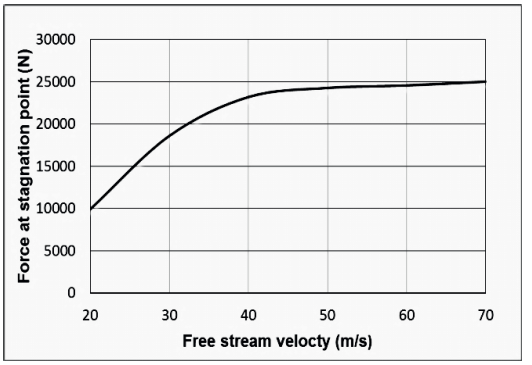


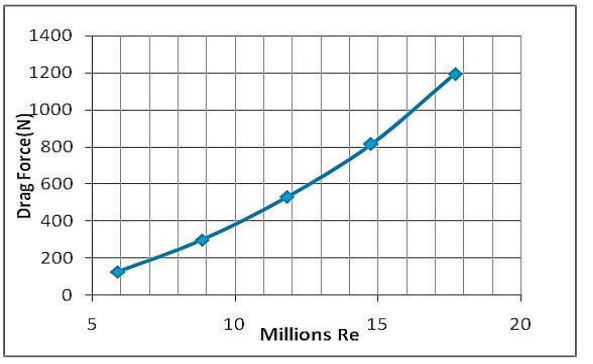
Model 1 model 2 model 3

2.Result

2.1 Effect of Velocity on Drag Force

Drag force is the result of the pressure difference between the front and the rear of the car. As the velocity of car increases, the stagnation pressure increases as shown in Fig. 1(a) and at the same time the pressure at rear decreases too due to the increase in momentum of air. So the difference between the pressure at stagnation or front and the pressure at rear of the car increases as the velocity and hence Reynolds no. increases. The relation between the drag force and Reynolds no. is as Fig.1(b). At the free stream velocity 50 m/s, the coefficient of drag CD is found to be 0.3233. Further analysis for drag reductions are done regarding this as a base value and compared the relative reduction of drag. .





2.2 Drag reduction by Rear Underbody Modification

When a car moves forward, a low pressure zone is created behind the car. This low pressure zone pulls the car from behind opposite to its moving direction and creates pressure drag. This low pressure zone is created due to the separation of flow and consequent vortices are generated at the back of the car. To reduce this pulling-back effect, a unique idea is to slice the rear under-body at certain angles which actually directs some flow from the under-body to the low pressure zone. This reduces the effect of vortices and low pressure effect. Another popular way to reduce the rear end separation is to use under-body diffuser which also adds elegance in aesthetics of the car, but it has less flow area as the rear under-body is not fully sliced out. So less reduction of drag is experienced then similar degree of rear under-body slicing.

modifications	Descriptions of modifications	C_D	ΔC_D	% of reduction of C_D
None	-	0.3233	-	-
Modification1	β =2.5°, rear under-body sliced	0.3083	0.01500	4.639
Modification2	β =5.0°, rear under-body sliced	0.2962	0.02707	8.373
Modification3	β =10.0°, rear under-body sliced	0.2694	0.05386	16.58
Modification4	β =12.5°, rear under-body sliced	0.2517	0.07156	22.13
Modification5	β =12.5°, under-body diffuser	0.2926	0.03070	9.5

Conclusion

- Flow separation is responsible for the major portion of aerodynamic drag of racing cars.
- The main design consideration to reduce the drag is to maintain a streamline shape, reducing surface roughness, fewer joints of the body or avoiding sharp fillets, controlling lift force, air or exhaust gas flow towards the low pressure zone at the rear portion of the car etc.
- These must be considered while designing a car for higher speed and acceleration as well as for better fuel efficiency and control
- Exhaust gas redirecting towards the low pressure zone at the rear of the car is effective to some extent
- About 3.3% drag coefficient can be reduced by this procedure considering the ideal exhaust gas composition.
- With the increase in frontal area also coefficient of discharge can be decreased
- Value of coefficient of drag decreases as the speed of exhaust gas decreases. Reason behind this is that higher velocity, exhaust gas induces the surrounding air to create a lower pressure, which elongate or enlarge the low pressure zone or reduce the pressure again and increase the drag
- Cutting out the section of firewall and providing wing at front end can reduce drag co-efficient from 0.85 for the standard race car to 0.70 for the modified car with front wing, whereas negative lift is increased from 0.2 for standard race car to 0.25 for the model 3.
- The pressure at firewall found to be reduced for the modified cars due to providing space to flow the air through cut out section, where flow remains attached and helps to decrease the drag. Thus overall pressure near the driver head region is reduced from 340Pa to 80 Pa. for the modified car with front wing
- Model having wing at the front end and having cut section at firewall shows less drag and lift, shows better aerodynamics characteristics than other two models.

References

[1]Toukir Islam, S.M. Rakibul Hassan and Dr. M. Ali, Aerodynamic Drag of Racing Cars, Global Engineering, Science and Technology Conference, Dhaka, Bangladesh, 2012

[2]Sneh Hetawal, Mandar Gophane,Bilandi Ajay, Yagnavalka Mukkamala, Aerodynamic Study Of Formula SAE Car,Global Congress On Manufacturing And Management ,GCMM 2014 Procedia Engineering 97 (2014) 1198 – 1207

[3]Md. MunirIslam, M.Mamun, Computational Drag Analysis over a Car Body, International Conference on Marine Technology, Dhaka,Bangladesh, 2010.

[4]BhaviniBijlani, Dr. Pravin P. Rathod, Prof. Arvind S. Sorthiya, Experimental and Computational Drag Analysis of Sedan and Square-Back Car, International Journal of Advanced Engineering Technology, 4(2013) 63-65.

[5] M. Rouméas, P. Gilliéron and A. Kourta, Drag Reduction by Flow Separation Control on a Car after Body, International Journal for Numerical Methods in Fluids, 60(2009) 1222–1240.