Proceedings of the Global Engineering, Science and Technology Conference 2012
28-29 December 2012, Dhaka, Bangladesh

Aerodynamic Drag of Racing Cars

Toukir Islam, S.M. Rakibul Hassan and Dr.M.Ali

Aerodynamic drag is the force opposite to the direction of motion that acts on a body moving through air. Drag force affects the performance and fuel efficiency of aircraft, road vehicles or racing cars as it works opposite to the direction of movement. About 50 to 60% of total fuel energy is lost only to overcome this force. Thus reduction of aerodynamic drag has become one of the prime concerns in vehicle aerodynamics and great efforts in research have been employed due to market competition for better fuel economy and performance of aircraft and road vehicles. For racing cars, this effort is thousand times intensified as speed and acceleration is the key to success in racing. A fraction of second can make a great difference between any two racing cars and the main reason of this difference is the drag force. This article focuses on types or components of aerodynamic drag that are significant for cars, measurement of drag co-efficient and suggestions regarding design considerations to reduce drag.

Field of research: Mechanical Engineering.

Keywords: Aerodynamic drag, separation, Reynolds number, coefficient of drag.

1. Introduction

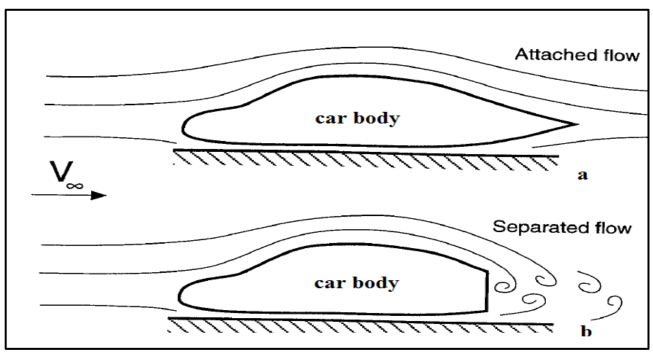
Drag force can be subsonic or supersonic depending on Mach number. But for vehicle aerodynamics, we mainly consider subsonic drag during design. Subsonic drag is mainly two types: profile drag, which is due to shape and finishing of car body and induced drag which is generated due to lift force. Now the profile drag can be divided into three different components- separation pressure drag, viscous or skin friction drag and interference drag. In conventional road vehicle, separation pressure drag is between 50 to 90 percent of total drag which depends on the proprieties of air flow. Airflow over a car can be attached or non-attached shown in Fig 1. For a completely attached flow over a car, the pressure acting on the rear surface gives rise to a force in the forward direction which completely counteracts the pressure acting on the front surface producing a force in the rearward direction, resulting in zero pressure drag. However, if the flow is partially separated over the rear surface, the pressure on the rear surface pushing forward will be smaller than the fully attached case (Katz, 1995).

The pressure acting on the front surface pushing backwards will not be fully counteracted. That flow is non-attached and it gives rise to a net pressure drag on the car. Due to flow separation, vortices generate low pressure field that tends to suck the

Toukir Islam, Room # 4007, Titumir Hall, BUET, Dhaka 1000, toukir.buet07@gmail.com
S.M. Rakibul Hassan, 16/5 khilgaon Bagicha,Dhaka 1219, rakibulhassan21@gmail.com
Dr.M.Ali, Professor, Department of Mechanical Engineering, BUET, mali@me.buet.ac.bd

car in the opposite direction of the flow, thus creating drag force. Skin friction drag is the consequence of frictional force between the body and air. Its magnitude totally depends on viscous effect, surface area, surface roughness and state of boundary layer (i.e. laminar or turbulent) thus difficult to measure theoretically because unlike the laminar flow situation, there are no exact analytical solutions for turbulent flow to measure the friction drag and analysis of any turbulent flow requires some amount of empirical data (Anderson, 2010). Induced drag is a by-product of lift or downforce. Lift or downforce occurs only because there is a pressure differential. For example, a higher pressure on the underside of a car than the top surfaces will cause lift. A wing with a higher pressure on the upper surface than the lower surface will develop downforce. Trouble is, these pressure differences always result in vortices of airflow between the two surfaces. The energy needed to create these vortices results in increased drag. Interference drag is that comes about because vehicles are not single bodies without appendages. And unfortunately, when different shapes are combined to form a practical vehicle, the total drag is always greater than the sum of the drag of individual parts. The flow over one surface interferes with the flow over another surface effectively.

Fig 1: (a) Attached flow; (b) separated or non-attached flow (Katz, 1995).



Reducing separation pressure drag by keeping the flow attached will result in a greater skin friction drag. Interference drag will always exist as it is impossible to produce a car shape that does not have appendages. Pressure drag contributes a large percentage in total aerodynamic drag where viscous, induced and interference drag contributes very small percentage but have to be considered important for race car design.

3. Literature review

To analyze or compare the drag between any two bodies, drag forces of the corresponding bodies are not used directly. Rather a dimensionless number is used which is the coefficient of drag (C_D) defined as,

$$C_D = F_D / (q_\infty \times S) \tag{1}$$

Here F_D is the drag force, S is surface area and q_∞ is the free stream dynamic pressure. The free stream dynamic pressure can be expressed as,

$$q_\infty = \frac{1}{2 \times \rho_\infty \times V_\infty^2} \tag{2}$$

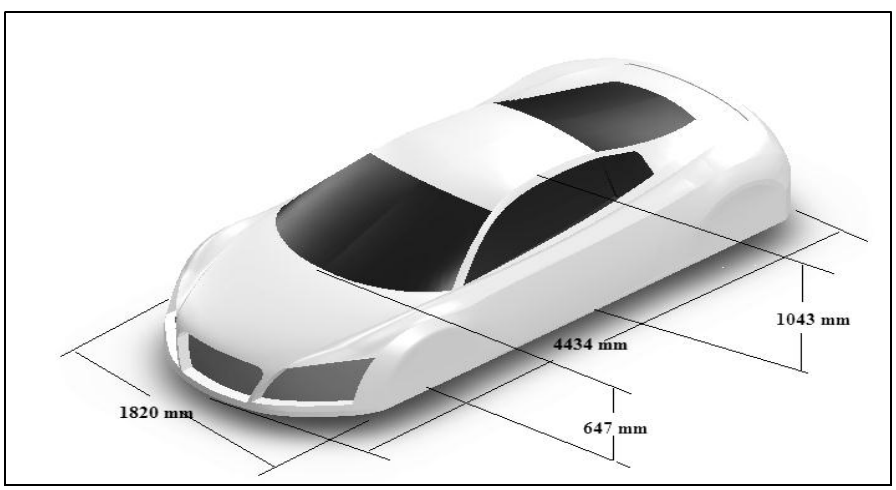
Here the free stream density ρ_∞ and velocity V_∞ are measured in the free stream flow and considered constant at any point in the free stream. But most daunting task is to consider the area *S* as it may be different for different types of drag and for different shapes. For pressure drag of racing cars, it is the frontal area to be used in calculations and using this area, C_D for car is usually in the range of 0.25 ~ 0.45 (Islam and Mamun, 2010). But for measuring the skin friction drag, it is the surface area that is used as S. Now F_D is in axial direction of the flow and if the flow direction is toward the x axis and the angle of attack α is zero, we can replace F_D by axial force N_x. So from equation 1 and 2,

$$C_D = Nx \,/\, (0.5 \times \rho_\infty \times V_\infty^2 \times S) \tag{3}$$

This is the equation used though out the article to calculate coefficient of drag. Here only the coefficient for pressure drag is measured using the frontal area and as the body is three dimensional; the induced and interference drag is included with that. Drag force changes with Reynolds number i.e. free stream velocity. If *L* is the length of the car and *μ* is the dynamic viscosity of air, the Reynolds number that defy the flow conditions can be expressed as,

$$Re = (\rho_\infty \times V_\infty \times L) \,/\, \mu \tag{4}$$

Fig 2: The car for analysis



2. Methodology: Numerical Procedure

Favre-averaged Navier-Stokes equations are used here, where time-averaged effects of the flow turbulence on the flow parameters are considered, whereas the other, i.e. large-scale, time-dependent phenomena are taken into account directly. Through this procedure, extra terms known as the Reynolds stresses appear in the equations for which additional information must be provided. To close this system of equations, Flow Simulation employs transport equations for the turbulent kinetic energy and its dissipation rate, the so-called k-ε model. Flow Simulation employs one system of equations to describe both laminar and turbulent flows and transition from a laminar to turbulent state or vice versa is possible. The conservation laws for mass, angular momentum and energy in the Cartesian coordinate system rotating with angular velocity Ω about an axis passing through the coordinate system's origin can be written in the conservation form as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} + \frac{\partial \mathrm{P}}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\tau_{ij} + {\tau_{ij}}^R\right) + S_i$$

$$\frac{\partial(\rho H)}{\partial t} + \frac{\partial(\rho u_i \mathrm{H})}{\partial x_i} = \frac{\partial}{\partial x_i}\left[u_j\left(\tau_{ij} + {\tau_{ij}}^R\right) + q_i\right] + \frac{\partial \rho}{\partial t} + S_i u_i - {\tau_{ij}}^R \frac{\partial u_i}{\partial x_i} + \rho\varepsilon + Q_H$$

$$H = \mathrm{h} + \frac{u^2}{2}$$

Here *u* is fluid velocity, ρ is fluid density, S_i is a mass-distributed external force per unit mass due to porous media resistance , a buoyancy (- ρg_i), and the coordinate system's rotation, *h* is the thermal enthalpy, Q_H is a heat source or sink per unit volume, τ_{ij} is the viscous shear stress tensor, q_i is the diffusive heat flux. The subscripts are used to denote summation over the three coordinate directions.

Now for Newtonian fluids, the viscous shear stress tensor is defined as,

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\,\delta_{ij}\frac{\partial u_k}{\partial x_k}\right)$$

According to Baussinesq assumption, the Reynolds-stress tensor has following form:

$${\tau_{ij}}^R = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\,\delta_{ij}\frac{\partial u_k}{\partial x_k}\right) - \frac{2}{3}\,\rho k \delta_{ij}.$$

Here δ_{ij} is the Kronecker delta function; equals to unity when i=j and zero otherwise. μ is the dynamic viscosity coefficient, μ_t is the turbulent eddy viscosity coefficient and *k* is the turbulent kinetic energy. Point to be noted that both *k* and μ_t are zero for laminar flow. In the frame of *k*-ε turbulence model, $\mu_t = f_\mu \frac{C_\mu \rho k^2}{\varepsilon}$; f_μ here is the turbulent viscosity factor; defined as , $f_\mu = \left[1 - \exp(-0.025\, R_y)\right]^2 \times [1 + \frac{20.5}{R_T}]$; where, $R_T = \frac{\rho k^2}{\mu\varepsilon}$ and $R_y = \frac{\rho k^{0.5}}{\mu} \times y$. Here *y* is the distance from the wall. This function of f_μ allows us to take into account laminar- turbulent transition. Two additional transport equations are used to describe the turbulent kinetic energy and dissipation at steady state,

$$\frac{\partial}{\partial x_j}(\rho u_i k) = \frac{\partial}{\partial x_i}[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}] + \tau_{ij}{}^R \times \frac{\partial u_i}{\partial x_j} - \rho\varepsilon + \mu_t P_B$$

$$\frac{\partial}{\partial x_j}(\rho u_i \varepsilon) = \frac{\partial}{\partial x_i}[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial k}{\partial x_i}] + C_{\varepsilon 1}\frac{\varepsilon}{k}\left[f_1 \times \tau_{ij}{}^R \times \frac{\partial u_i}{\partial x_j} + \mu_t C_B P_B\right] - C_{\varepsilon 2} f_2 \frac{\rho\varepsilon^2}{k}$$

Here, $f_1 = 1 + (\frac{0.05}{f_\mu})^3$, $f_2 = 1 - \exp(-R_T{}^2)$, $P_B = -\frac{g_i}{\sigma_B} \times \frac{1}{\rho}\frac{\partial P}{\partial x_i}$

Now for these equations C_B is unity when $P_B > 0$ and zero otherwise. $\sigma_B = 0.9$, $C_\mu = 0.09$, $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2}$=1.92, $\sigma_k = 1$, $\sigma_\varepsilon = 1.3$. These values are found empirically. These equations describe both laminar and turbulent flow. (Technical reference, 2012) The computational domain is of 17.852 m × 6.012m × 3.7037m. As no wheels are used, some clearances are kept from the ground according to wheel diameter to ensure flow though the under-body of the car. The free stream pressure is considered atmospheric where the temperature is 20°C. The working fluid is air with density 1.21 kg/m^3 at free stream. External flow is considered with adiabatic wall and the entire numerical calculation is done for steady state conditions.

3. Findings and Discussions:

Model used for this analysis resembles a racing car, AUDI r8 except for the side skirts. For the measurement of separation and pressure drag, it is convenient to take the frontal area of the car which is 1.68 m^2 for the model used for analysis. Now for free stream velocity of 50 m/s when surface roughness is not considered,

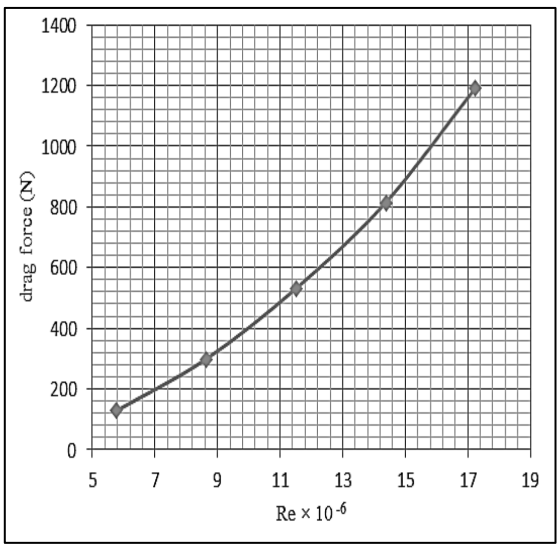
Table 1. Coefficient of drag for flow velocity 50 m/**s**

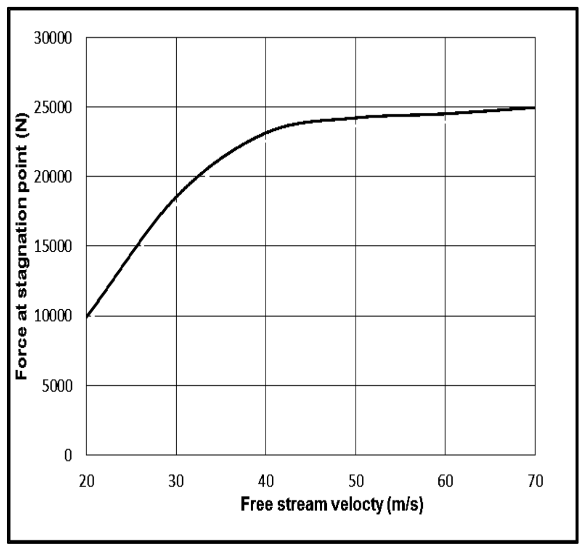
Parameters	*magnitude*
Density of air at inlet	1.21 kg/m^3
Dynamic pressure at inlet	1512.5 Pa
Pressure Drag force	814.8136 N
Coefficient of Drag C_D	0.3233

Here C_D for free stream velocity found to be 0.3233 which is lower than the actual C_D of AUDI r8 model. This difference may be due to- (a) this analysis is only for steady state conditions, (b) the velocity at the inlet is considered to be one directional which is not desirable in race-track conditions, (c) no wheels are used with the model whereas wheels are one of the major sources of aerodynamic drag. Fig 3 shows the non-liner variation of C_D with respect to Re. with increasing Re, the free stream velocity increases but the drag force increases more rapidly due to larger pressure at stagnation point, more separation, shift of laminar boundary layer to turbulent and vortices at the rear of the car.

Fig 3: Drag forces at different Re

Fig 4: Stagnation force at different V_∞.





At the stagnation point, drag force towards the flow direction is maximum. It reduces in fluctuating manner, presented in Fig 5. This fluctuation is largely due to local wakes and vortices caused due to flow separations and it is affected by the sudden change of body profile of the model. Higher the force at stagnation point, higher the value of C_D. Vorticity contours in Fig 7 resemble the effect of sudden changes of flow directions due to the profile of the model. Separation of flow at the rear of the car reduces the pressure to vacuum in some rejoins that tends to pull the car in backward direction. These suction form rear and stagnation pressure from the front dominate the percentage of pressure drag.

Fig 5: Drag force variation at 50 m/s

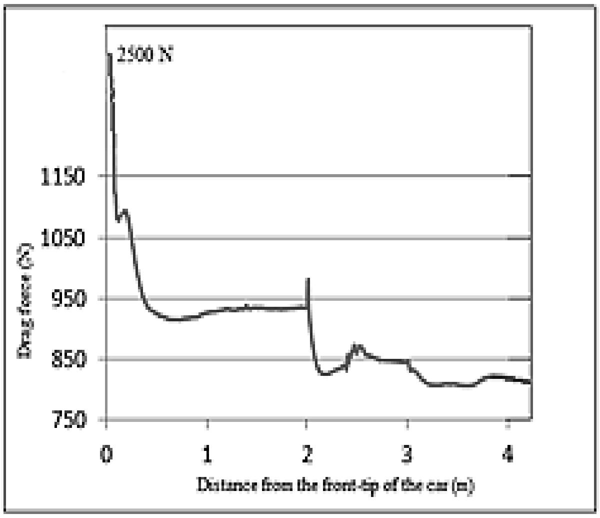


Fig 6: Mach number variation at 50 m/s

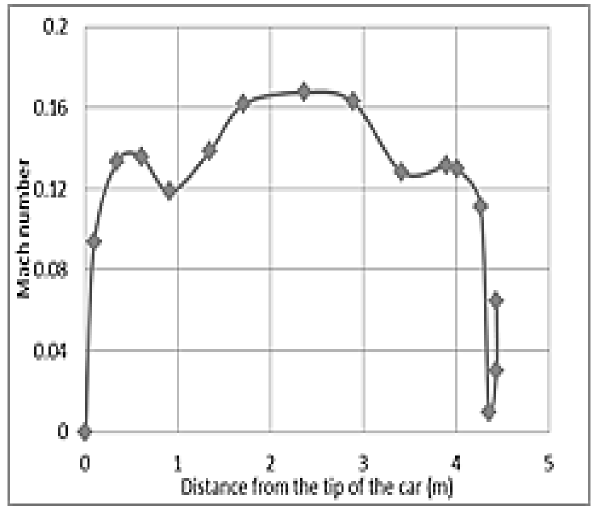
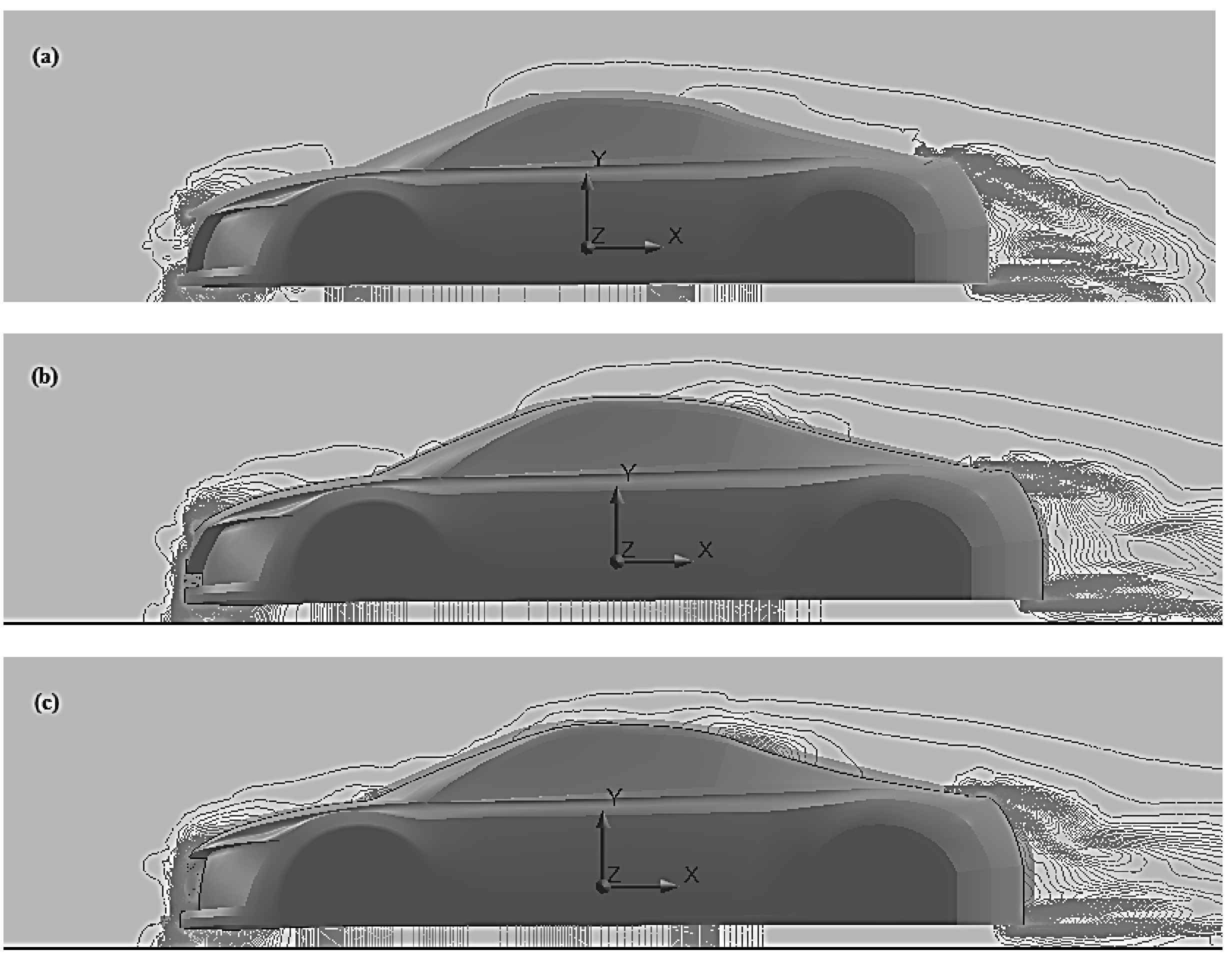


Fig 7: Vorticity isolines at 50 m/s at (a) centerline (b) 0.4m offset (c) 0.5m offset in z direction



4. Conclusion:

Different types of aerodynamic drag and influences of several parameters on the aerodynamic drag have been discussed through this article. This article should be helpful to understand the drag phenomena and procedure to measure it numerically. The main design consideration to reduce the drag of any bluff should be- “keep the flow attached to the body as much as possible”. That means maintaining 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 zones at the rear portion of the car etc. should be considered while designing a car for higher speed and acceleration as well as for better fuel efficiency and control.

5. References:

[1] Anderson, John D.,Jr, Fundamental of Aerodynamics [In SI unit], Fifth Edition, NEW DELHI : Tata McGraw Hill Education Private Limited, 2010.

[2] Islam Md. Munir , M. Mamun. “Computational Drag Analysis over a Car Body ”. International Conference on Marine Technology, Dhaka, 2010.

[3] Katz Joseph. “Race Car Aerodynamics- Designing for Speed”. Cambridge: Bently Publishers,1995.

[4] SolidWorks Flow Simulation 2012, Technical Reference.