Proceedings of the
International Conference on Mechanical Engineering and Renewable Energy 2017
(ICMERE2017) 18 – 20 December, 2017, Chittagong, Bangladesh

ICMERE2017-PI-235

Effect of Side Taper on Aerodynamic Drag of a Simple Body Shape

Mohammad Mashud[1], K. M. Mohiuddin Abeer[2], Md Nazmul Hasan[3] and Md Enamul Haque[4]

[1-2]Department of Mechanical Engineering, Khulna University of Engineering & Technology, Khulna-9203, Bangladesh
[3]Department of Mechanical Engineering, Shonargaon University of Science & Technology, Dhaka-1215
[4]Faculty of Engineering, World University of Bangladesh, Dhaka-1215
mdmashud@yahoo.com*, mohiuddinabeer8@gmail.com, nhasankuet@gmail.com, enam.kuet.me.2k10@gmail.com

Abstract-*Nowadays the world is moving so fast. Still the necessity of a faster vehicle is ever increasing. But simultaneously the price of fuel is also increasing along with threats to the world like global warming. So, it has become a challenge for the automotive design engineers to design such cars which use less fuel still providing with more speed and an aesthetic view. The main obstacle in this case is the aerodynamic drag which pulls back the vehicle resulting in less speed and more fuel consumption. So, the design engineers are leaving no stone unturned to find a suitable way out from all aspects to reduce this aerodynamic drag. There are different techniques to reduce aerodynamic drag of a vehicle. Side tapering is one of the methods to reduce drag which is a simple modification of the body. In this thesis, instead of using very simple shape of a model, a more relevant simple car like shape resembling a hatchback vehicle is utilized. The model is tapered on both sides along the whole height by* 5^0, 10^0, 15^0 *and* 20^0 *one by one with a constant taper length of 14% of the total length. By testing the model in the wind tunnel, it was found that* 10^0 *was the optimum taper angle which can be helpful in minimizing the aerodynamic drag. Above* 10^0 *the drag again starts increasing.*

Keywords: Vehicle Aerodynamics, Wind tunnel, side taper

1. INTRODUCTION

The rapidly increasing fuel prices and the regulation of greenhouse gasses to control global warming give tremendous pressure on design engineers to enhance the current designs of the vehicle using the concepts of aerodynamics as to enhance the efficiency of vehicles [1]. Drag reduction of a vehicle is one of the desired objectives of vehicle manufacturer as it leads to reduction in fuel consumption and improved acceleration with lower power [3]. Vehicle wake is one of the major contributors to its drag. Fuel consumption due to aerodynamic drag consumed more than half of the vehicle's energy [2]. Thus, the drag reduction program is one of the most interesting approaches to cater this matter. Aerodynamic drag consists of two main components: skin friction drag and pressure drag. Pressure drag accounts for more than 80% of the total drag and it is highly dependent on vehicle geometry due to boundary layer separation from rear window surface and formation of wake region behind the vehicle [4][7]. The location of separation determines the size of wake region and consequently, it determines the value of aerodynamic drag. According to Hucho [1], the aerodynamic drag of a road vehicle is responsible for a large part of the vehicle's fuel consumption and contributes up to 50% of the total vehicle fuel consumption at highway speeds[5][8]. Reducing the aerodynamic drag offers an inexpensive solution to improve fuel efficiency and thus shape optimization for low drag become an essential part of the overall vehicle design process [2]. It has been found that 40% of the drag force is concentrated at the rear of the geometry [3].

There are numerous techniques to control and reduce vehicle wake in order to reduce its drag. Tapering sides of passenger vehicle is one such simple technique which can be practically implemented on commercial and passenger vehicles [4][9]. Proper study of aerodynamic behavior of this side tape/boat tail feature is necessary. Also, it is helpful to have a design rule that can be applied with correct understanding of physics behind it. If range of best angles and lengths for boat tail can be found then early provisions in exterior design can be made to properly implement this feature without affecting other constraints on a passenger vehicle shape[6]. Boat tails are heavily studied for truck trailers. However, boat tails on trailers come as an extension and generally all four sides of rear of trailer gets boat tail extensions. Side tapering is not preferable on trailers as it would reduce load carrying volumetric capacity. There is slight difference in approach and construction for boat tail and side tapering. Side tapering is generally done to modify sides of vehicle to reduce its cross-sectional area at the rear. Boat tailing is mostly performed using extensions on existing rear end of vehicle[6].

Many researchers have reported significant drag reduction using angled boards creating cavity and boat tail effect on scaled models of simplified real or close to real trucks. Cooper [4] reported optimum angle as 15^0

with 3 side boat tail. Schoon and Pan [5] found 12.5^0 as optimum angle with 4 sided boat tail Studies are also performed on generic bodies to asses boat tail drag reduction potential such as Mair [6] and also Mair and Wang [7]. Gilhaus [8] studied simple rectangular bodies representing bluff body vehicles near ground and with angled boards at rear end to provide cavity and boat tail effects. Howell et al [9] studied taper angles on sides and top of Windsor body. Authors found that for short taper lengths initially give sharp drag reduction but this trend is reversed and increase in drag occurs for angles greater than 10^0. For large taper angles drag shoots up for this short taper length. Mostly researchers worked on scaled generic models and found different combinations of lengths and angles for best drag reduction. It is therefore necessary to have a database for full scale model which has most of the features of real world passenger vehicle which can serve as ready reckoner design rule. Hence a side taper study is performed in this work to generate a database and a guideline giving best angle and length for drag reduction.

In this thesis, the modification is taken as taper on two sides and the taper length is about 14% of the whole length. The taper angle is varied as 5^0, 10^0, 15^0 and 20^0. For all these taper angles experiment is carried out to investigate on the change of drag with taper angles.

2. METHODOLOGY

The experiment is to be carried out in the wind tunnel tester with a wooden model of a simple body shape resembling a hatchback. Here, the drag co-efficient on the body is to be determined from pressure co-efficient for different taper angles 5, 10, 15 and 20 degrees respectively. The readings are taken at 3 different wind velocities such as 29.2434 m/s, 34.1984 m/s and 39.1568 m/s. For this purpose, firstly the taper length is chosen as about 14% of the total length. Then the required taper angle is made. The drag equation is a formula used to calculate the force of drag experienced by an object due to movement through a fully enclosing fluid. The formula is accurate only under certain conditions: the objects must have a blunt form factor and the fluid must have a large enough Reynolds number to produce turbulence behind the object. The equation is:

$$F_D = \frac{1}{2}\rho u^2 C_D A \quad [8]$$

Here,

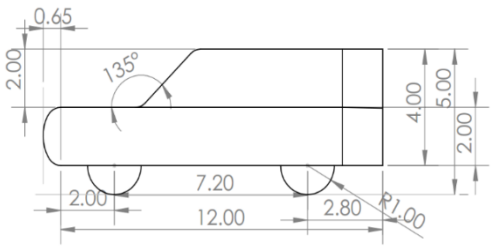
F_D is the drag force, which is by definition the force component in the direction of the flow velocity ,

ρ is the mass density of the fluid,

u is the flow velocity relative to the object,

C_D is the drag coefficient – a dimensionless coefficient related to the object's geometry and taking into account both skin friction and form drag, and

A is the reference area.

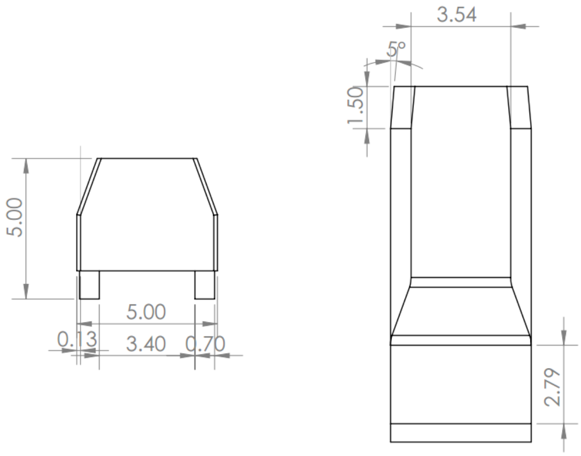


3. MODEL DESIGN AND CONSTRUCTION

In this thesis, a simple body shape resembling ahatchback car is to be used. So, for designing the model firstly the real dimensions of a hatchback car (here used Mitsubishi Pajero) are taken. Then by scaling the dimensions by almost 20 times smaller the model was drawn using CAD software. The scaling is not strictly maintained as the body shape is to be simple i.e. without any complex shape. The dimensions are in inch. The CAD model is illustrated below:

Figure 1: Left side view with dimensions (in inch).

Figure 2: Back and top view with dimensions (in inch).



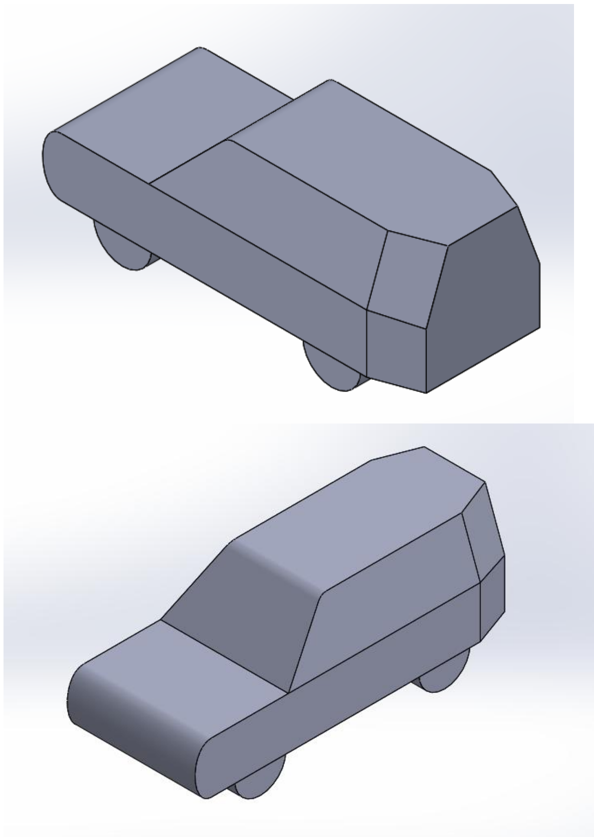


Figure 3: Isometric view of CAD model with side taper.

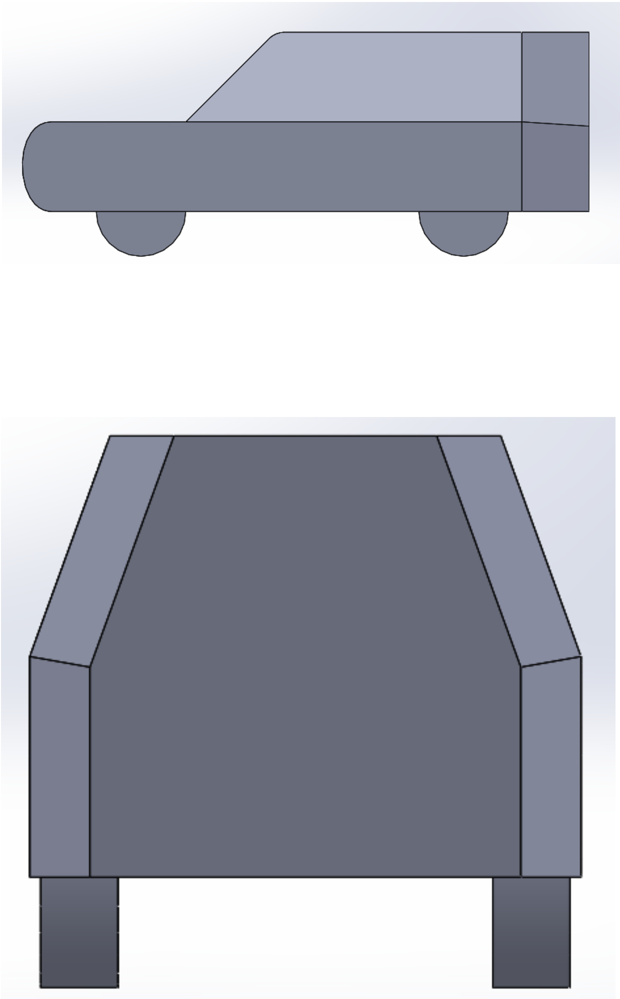


Figure 4: Left side and back view of CAD model.

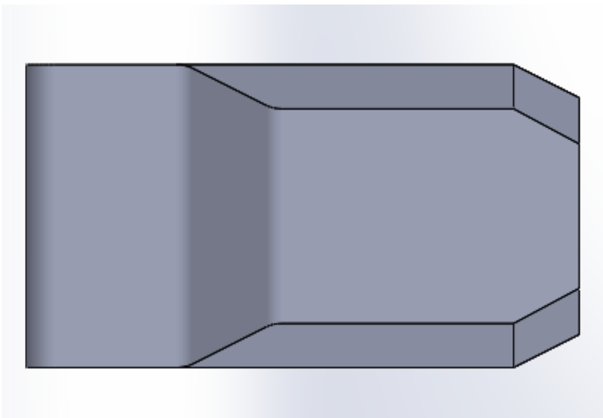


Figure 5: Top view of the CAD model.

For the experiment to be carried out in the wind tunnel a wooden model of the simple body shape is crafted. The material used is wood. The dimensions are used same as the CAD model. The model is as below:

Figure 6: Left side view of the constructed wooden model.

Figure 7: Back and top view of the constructed wooden model.

Figure 8: Isometric view of the constructed wooden model.

4. RESULT AND DISCUSSION

The vehicle model was investigated into the wind tunnel of the Aerodynamic Lab of the Khulna University of Engineering & Technology. Firstly, the pressure distribution over the body was calculated and then the drag co-efficient was calculated. The result is shown below,

4.1. Pressure distribution

From the three graphs shown below it is clear that the nature of all these graphs are almost same. Firstly, when the air strikes at the front of the car the pressure increases. This phenomenon can be explained by using the Bernoulli's theorem. When the air strikes at the front the velocity of air in the direction of flow becomes zero and it is deflected by a right angle. So, here the pressure becomes maximum. Then when the air passes over the bonnet of the car the air gets separated from the surface due to the formation of bubbles and so, a low-pressure region is created here. Then the air strikes at the windshield of the car. So, the pressure is again increased. While passing over the roof of the car the pressure is more or less constant. But for 20^0 taper angle the pressure at the end is abnormally high. This uneven distribution of pressure occurs due to the increased taper angle.

For Re = 7.31×10^5:

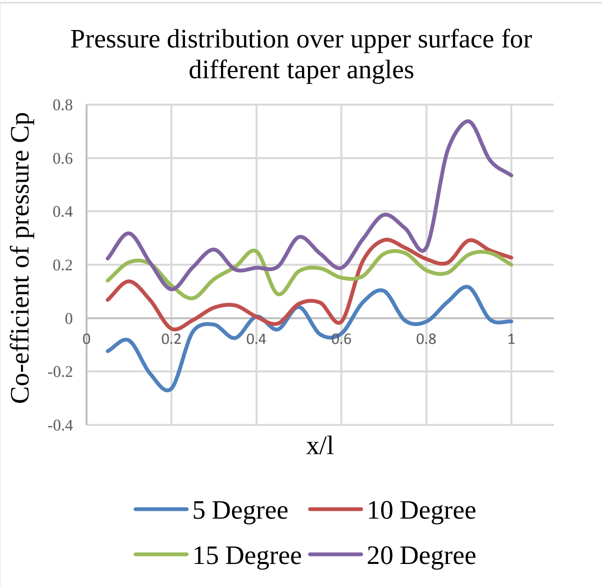


Figure 9: Coefficient of pressure vs Fraction of length at Re = 7.31×10^5.

For Re = 8.55×10^5

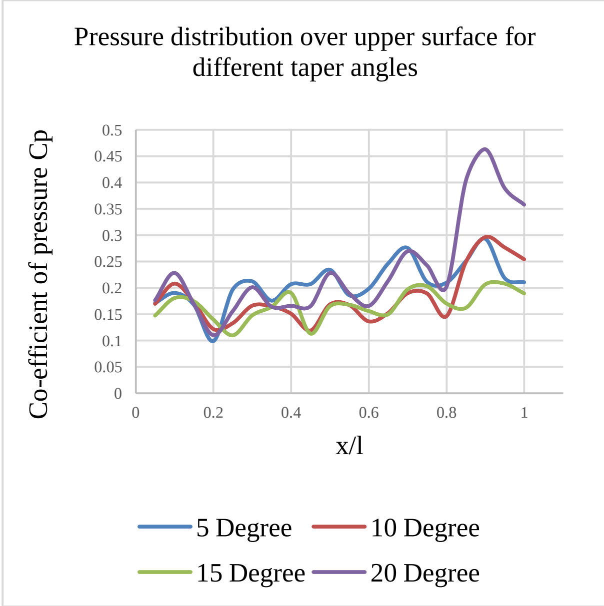


Figure 10: Coefficient of pressure vs Fraction of length at Re = 8.55×10^5.

For Re = 9.78×10^5

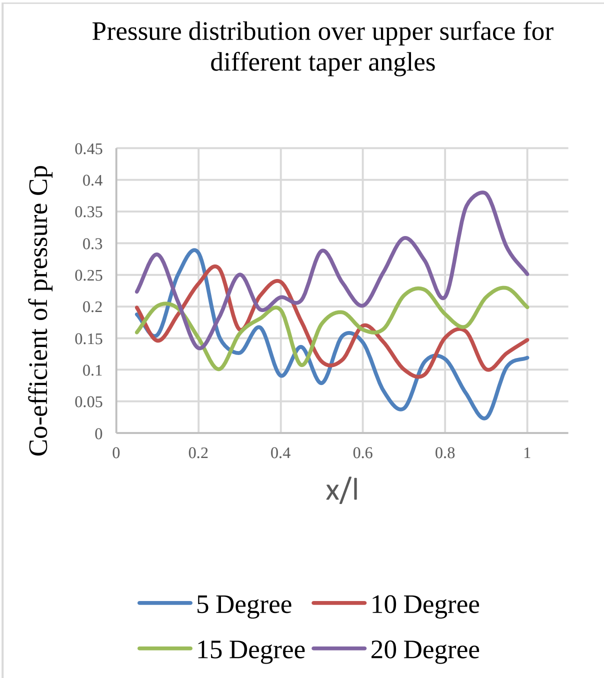


Figure 11: Coefficient of pressure vs Fraction of length at Re = 9.78×10^5.

4.2. The coefficient of Drag

The main objective of this thesis was to reduce the coefficient of drag of the automotive vehicle model by using the side tapering method. The experimental result is agreed with the theoretical knowledge that this can reduce the drag coefficient. So, the drag co-efficient is now calculated. The distribution of drag co-efficient over the entire body for different side taper angles is compared below in a single graph:

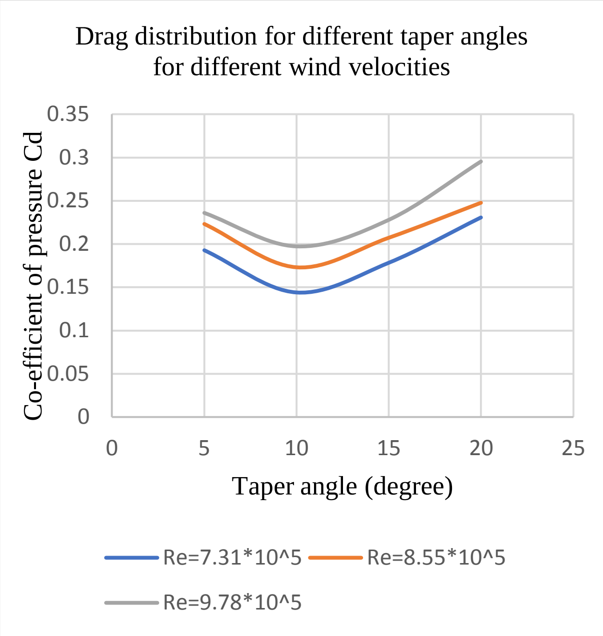


Figure 12: Drag distribution for different taper angles for different wind velocities.

Here, we see that for all three velocities the drag co-efficient firstly decreases from 5 degree to 10 degree. But then C_d starts increasing for 15 and 20 degree of taper angles.

This is because a low-pressure swirl for 10^0 is created little farther than other cases from base of the vehicle. This low-pressure zone acts like center of vortex behind the base of a vehicle. This low-pressure zone must be as far away as possible from base so that its effect on base surface is lesser. This is one of the reasons for 10^0 case to have lower drag. Also, the pressure distribution in wake for other than 10^0 case is not properly distributed which cause uneven upper and lower lobe portion of the wake. Wake structure forms two distinct lobes as taper angle increases. These two lobes become more pointed as taper angle is increased; also, turbulent kinetic energy is seen to be increasing at top portion of these two lobes. For all lengths, wake behavior for particular angle and taper length is similar. It is also observed that as angle increases, wake distribution at the back of vehicle becomes uneven and top side portion extends much larger than bottom portion. Initially for small angles of 5^0 and 10^0, tapering directs lesser flow from model sides, into the wake. As angle increases, all flow along sides is directed into the wake but then diffuser also directs underbody flow upward. Hence most of the sideward flow is lifted up due to diffuser which results in extension of top portion of wake and curtailing of bottom portion.

It is most important to learn here that even though it is expected to decrease drag as side taper angle increase but it does not happen after 10^0 angle. Decrease in wake size will result decrease in drag. This is true only till 10^0 angle but after that two counter rotating strong vortices are formed which consume more energy in wake. Hence drag increases instead of reducing after 10^0 angle. Strength of vortices increases after 10^0 angle. After 10^0 angle, two counter rotating vortices formed, increase in magnitude and spread. Increase in magnitude of vortices results in more loss of energy in wake which in turn increases drag. Hence drag benefits for angles above 10^0 start reducing.

5. CONCLUSION

The following conclusions can be drawn from the above calculations:

- As the side taper angle increases, the drag gradually reduces up to 10 degree.
- Beyond 10 degree, as the taper angle increases the drag starts increasing instead of decreasing.
- So, the optimum taper angle is 10 degree with a taper length of 14% of total length.

6. ACKOWLEDGEMENT

The authors like to express gratitude and respect to Dr. Mohammad Mashud, Professor, Department of Mechanical Engineering, Khulna University of Engineering & Technology, Khulna, Bangladesh, for his support and guidance.

7. REFERENCES

[1] Hucho, W. H. and Sovran, G. (1993) Aerodynamics of road vehicles, Annual Review of Fluid Mechanics, Vol. 25(1), pp485-537.

[2] Mayer, W., and Wickern, G. (2011) The new audi A6/A7 family aerodynamic development of different body types on one platform, SAE International Journal of Passenger Cars-Mechanical Systems, Vol. 4(1), pp197-206.

[3] Chainani. A, and Perera. N, (2008) CFD Investigation of airflow on a model radio control race car, WCE 2008, 2-4July, London.

[4] Cooper, K., "The Effect of Front-Edge Rounding and RearEdge Shaping on the Aerodynamic Drag of Bluff Vehicles in Ground Proximity," SAE Technical Paper 850288, 1985, doi:10.4271/850288.

[5] Schoon, R. and Pan, F., "Practical Devices for Heavy Truck Aerodynamic Drag Reduction," SAE Technical Paper 2007-01-1781, 2007, doi:10.4271/2007-01-1781.

[6] Mair,W.A., Reduction of base drag by boat tailed afterbodies in low speed flow. Aeronautical Quarterly Vol.20, pp307-320,1969

[7] Wong,D.T-M.,Mair,W.A.,Boat-tailed afterbodies of square cross section as drag reduction devices,Vol.12,pp229-235,1983

[8] Gilhaus, A., Aerodynamics of heavy commercial vehicles. Vehicle aerodynamics short course 84-01, Von Karman Institute for Fluid Dynamics, Jan 1984.

[9] Howell, J., Passmore, M., and Tuplin, S., "Aerodynamic Drag Reduction on a Simple Car-Like Shape with Rear Upper Body Taper," SAE

Int. J. Passeng. Cars - Mech. Syst. 6(1):52-60, 2013, doi:10.4271/2013-01-0462.

8. NOMENCLATURE

Symbol	Meaning
F_D	Drag Force
C_P	Co- efficient of Pressure
C_D	Co-efficient of Drag
ρ	Mass density of fluid
u	Velocity of steam
Re	Reynolds Number