Available online at www.academicpaper.org

Academic @ Paper

ISSN 2146-9067

International Journal of Automotive Engineering and Technologies

Vol. 4, Issue 3, pp. 146 – 151, 2015

International Journal of Automotive Engineering and Technologies

http://www.academicpaper.org/index.php/IJAET

Original Research Article

Drag Coefficient Determination of a Bus Model Using Reynolds Number Independence

Hamit Solmaz[1*], Yakup İçingür[1]

[1] Gazi University, Faculty of Technology, Automotive Engineering Department, Ankara, Turkey

Received 09 July 2015 Accepted 21 September 2015

Abstract

Full-scale wind tunnel tests require too much time and cost to determine drag coefficients of vehicles. In this study, drag coefficient of a blunt bus model was tried to be determined using Reynolds number independence. A low speed wind tunnel having a free flow velocity is 28 m/s and has a rectangular cross section of 292 mm high and 292 mm wide, was used in experiments. The flow around the bus model was simulated with ANSYS CFX at wind tunnel conditions. As a result, aerodynamic drag coefficient of the bus model was determined as 0,66 and 0,65 according to results of CFX and experiment respectively after Reynolds number 57000. It is determined that the drag coefficient of the blunt vehicles can be determined by using low speed wind tunnel and Reynolds number independence.

Keywords: Drag coefficient, Reynolds number, vehicle aerodynamic, drag force, performance

*Corresponding Author:
E-mail: hsolmaz@gazi.edu.tr

1. Introduction

Aerodynamic structure and the flow around the road vehicles have been investigated for a long time. Significance of the aerodynamics is obvious, since it affects the fuel consumption, wind noise, and vibration [1]. It is known that a 10% drag reduction leads to approximately a 5% reduction of the fuel consumption of a bus at a common highway speed [2]. Land transportation is the most widely used passenger and freight transportation in Turkey and 95% of total of passenger transport performed by road. An estimated total savings of $100 million per year can be recognized in Turkey alone for just a 5% reduction in fuel use in intercity passenger transportation by buses. Detailed knowledge of the aerodynamic characteristics of passenger vehicles could lead to find out new solutions to reduce fuel consumption and emissions and improving the vehicle performance and passenger comfort [3]. Wind tunnel tests are used to determine aerodynamic characteristics of road vehicles [4]. Wind tunnel tests can be cheaper and easier than the road test, because smaller models can be used in wind tunnel tests. However, a full-scale wind tunnel test can be more costly than a road test [5]. Wind tunnel tests began with small-scale models. Experiments with small-scale models has advantages because it's more cheaper and easier than the full-scale tests [6]. However it is very difficult to match Reynolds number between prototype and model in small-scale procedure [7]. The test conditions must be established in such a way that the related forces between the model and the full-scale prototype must be scaled by a constant factor. Only when these conditions between the flows are established, data obtained from the wind tunnel test may be related quantitatively to the prototype flow. To ensure the dynamic similarity between the flows, dimensionless analysis show that the Reynolds numbers in both cases must be the same. In the case of predicting the drag force on a sphere, the results obtained from the model flow can be related to the prototype flow by using the relation of dynamic similarity [8].

$$\mathrm{Re}_{\mathrm{m}} = \mathrm{Re}_p \tag{1}$$

$$\left(\frac{\rho V L}{\mu}\right)_m = \left(\frac{\rho V L}{\mu}\right)_p \tag{2}$$

$$V_m = V_p \left(\frac{\mu_m}{\mu_p}\right)\left(\frac{\rho_p}{\rho_m}\right)\left(\frac{L_p}{L_m}\right) \tag{3}$$

As it seen from equation 3 in order to match the Reynolds numbers, as the model gets smaller in size, the required air speed inside the test section increases. According to equation 3 the maximum air speed required in test section would be 1920 km/h, for 80 km/h simulation with a 1/24 scale model. It's impossible to reach this speed in a wind tunnel. But there are several were to match Reynolds number for low speed wind tunnel. Wiedemann and Ewald were stated that the Reynolds number can be increased in a low speed wind tunnel by increasing turbulence ratio in the tunnel [9]. Chuan and Tao were built a low speed pressurized wind tunnel to increase Reynolds number [10]. Using these methods to increase Reynolds number can be effective but more expensive than ordinary wind tunnel tests. Instead of trying to increase Reynolds number in the wind tunnel, using the Reynolds number independence may be easier, cheaper and faster.

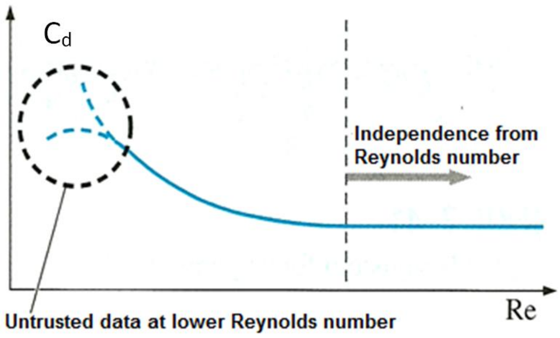

Figure 1. Reynolds number independence [11]

While the drag coefficient (C_d) is a strong function of Reynolds number at low Reynolds values, it is fixed and does not change after a certain Reynolds value (Fig. 1). The sharp-edged bodies, which tend to cause flow separation regardless of the character of the boundary layer, are insensitive to the Reynolds number. Therefore, determining the drag coefficient

of blunt bodies such as truck and bus, Reynolds number independence can be use [11–15].
The goal of this study is to determine drag coefficient of a bus model in a low-speed wind tunnel. The bus model I, which was tested in a water tunnel by Gürlek [3], has been used. The points where separation of flow occurs are determined analyzing the flow around the autobus with ANSYS CFX. At the end of the study, the results of analysis and test are compared.

2. Experimental Setup

2.1. Model Description

In the present experiment Gürlek's bus model I [3] has been studied. Figure 2 shows the bus model I geometry. The length of the model L=175mm, the height, H=66 mm and the width, W=56 mm.

$$l_c = \frac{6V_b}{A_b} \quad (4)$$

Bus models are longer vehicles than passenger cars. While determining Reynolds number vehicle length, width or height can be used as the characteristic length. However, none of the length of the model cannot characterize the model alone. Therefore, in this study characteristic length of the bus is determined due to the volume and surface area of the model. Using the equation 4 characteristic length of the bus model I have been determined as $l_c = 0,0765m$. Frontal area of the model I is $A_f = 0,0037m^2$.

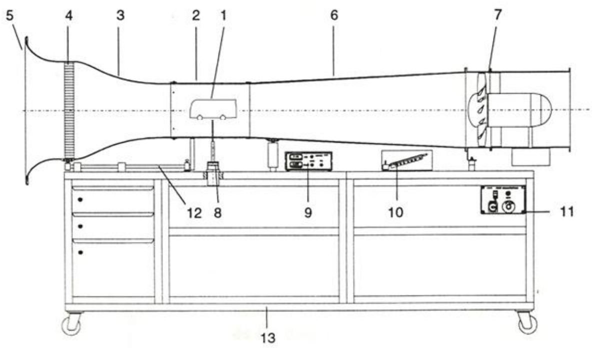

Figure 3. Schematic representation of wind tunnel.
1. Model 2. Test section 3. Nozzle 4. Flow rectifier
5. Inlet hopper 6. Diffuser 7. Fan 8.Force transducer
9. Amplifier 10. Tube manometer 11. Switch box
12. Guide 13. Trolley

2.2. Wind Tunnel

Figure 3 shows the wind tunnel that has used in experiments. The wind tunnel is an open type and subsonic wind tunnel. Maximum free flow velocity is 28 m/s and has a rectangular cross section of 292 mm high and 292 mm wide. Flow velocity in the test section can be measured by slant tube manometer. Velocity can be controlled from 0 m/s up to 28 m/s by switch box. Model has been placed center of the cross section to prevent the boundary layer development.
To measure the drag force acting on model I, load cell that shown number 8 has been used. Measurements have been performed with 2 m/s velocity increases between 2-27 m/s. Force measurements have been repeated five times for each flow velocity and averages has been taken to improve the accuracy. In order to calculate the drag coefficient of bus model equation 5 has used:

$$C_d = \frac{F_d}{\frac{1}{2}\rho A V^2} \quad (5)$$

where C_d is the aerodynamic drag coefficient, V is the flow velocity in cross section, F_d is the drag force acting on model, ρ is the density of air and A is the frontal area of the model.

2.3. Numerical Simulation

Reynolds number independence can be obtained when flow separation takes place without effects of the boundary layer. Therefore, the flow around the model should be examined. According to the theory of fluid mechanics, performs numerical simulation of 3-D flow field around the automotive, using the Navier - Stokes governing equations with $k-\varepsilon$ turbulence model [16-18]. ANSYS CFX has been used for the numerical simulation. The dimensions of the numerical wind tunnel were equal to the experimental wind tunnel. Boundary conditions set as: velocity inlet has been taken the same velocities in experimental research, pressure inlet and outlet has been equal to 1 atm because of the open wind tunnel.

3. Results and Discussion

Figure 4 shows streamlines around the bus model on side view. Flow has separated up and down on front side of the bus. As a result of the radius on the front roof section of the bus flow hasn't separated in this region. However, cornered front bumper structure of the bus has caused flow separation. At the back of the model a vortex region that's nodal point is closer to road has taken place.

Figure 4. Flow lines around the model (27m/s side view)

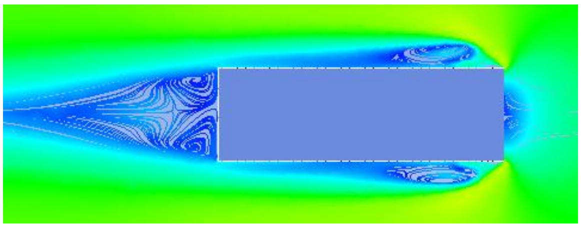

Figure 5. Flow lines around the model (27 m/s top view)

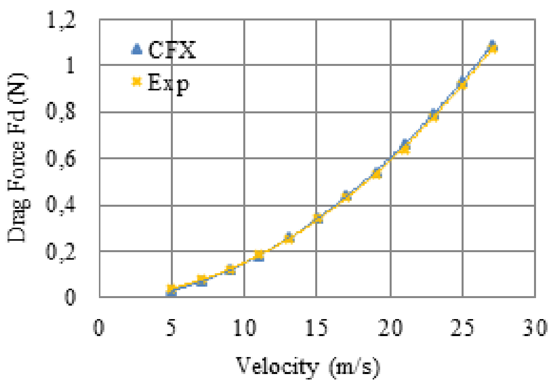

Figure 6. Drag force change by flow velocity

Figure 5 shows flow lines around the bus on the top view. Flow has driven on the front left and right sides and has been separated after the front edges of the bus. After the flow separation on front edges of the bus, flow has not been reattached again with the surfaces along the bus Contrary to side view, at the back side of the bus two vortex formations has been seen due to cornered side surfaces. It is seen in each two figures, the flow separation is formed only due to model geometry. Flow separation around the model is not formed due to boundary layer effects.

Figure 6 shows variations of drag force affecting the bus model in different wind speed. It is seen from the graphic the drag force values that obtained by experimental and CFX analysis results are too close to each other. However, in the range of 5-11m/s speed the drag force is lower in experimental result than CFX analysis. If it is accepted the density value does not change, only the air velocity and the drag force affects the drag coefficient. Small deviations in the drag force can greatly affect the drag coefficient especially at low speeds. The difference in drag force between CFX and experimental results at 5 m/s speed is 0,01 N.

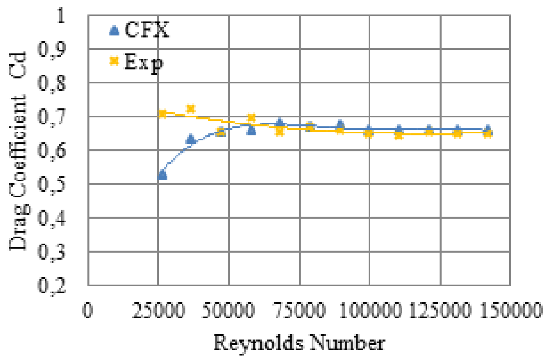

Figure 7. Drag coefficient change by Reynolds number

Figure 7 shows the drag coefficient values that obtained according to the results of CFX and experiment. At low Reynolds numbers the C_d values has changed in both the test and the CFX results. The whole range of Reynolds number values it is expected to C_d have a fixed value. If the Reynolds number of the model and prototype don't match, C_d values of the model can not be fixed. As shown in figure 7, the C_d value has been fixed by increasing the wind speed. According to results of both CFX and the experiment, C_d value has become to stable around Reynolds number 57000. Aerodynamic drag coefficient has started fixing at low Reynolds values due to blunt bus model. This situation is expected when objects do not have streamlined structure and drag coefficient remains constant after critical point of Reynolds number because of bad aerodynamic characteristics and blunt geometry of bus model. [6]. According to experimental and CFX results, Reynolds number independence has been provided. After the Reynolds number value 57000, according to the CFX and experimental results the drag coefficient of the model has been determined as 0,66 and 0,65 respectively. Gürlek [3] was determined the aerodynamic drag coefficient of the Model I as 0,62 in a water tunnel. The difference of the results between Gürlek [3] and this study could be resulted from the difference between air and water properties and small difference between models in production.

4. Conclusions

In this study, a bus model which height 66 mm, weight 56 mm and length 175 mm, was tested in a wind tunnel that's free flow speed 28 m/s and cross section 292x292mm^2, and test results was compared with CFX results. Using Reynolds number independence aerodynamic drag coefficient of the bus model was tried to be determined.

As a result, according to both the CFX and the wind tunnel test results aerodynamic drag coefficient of the bus model was fixed after the value of the Reynolds 57000. Because of the flow separations around the bus, aerodynamic drag coefficient value was stable at low Reynolds number. At the bottom, rear and each side of the bus, flow was separated naturally without any boundary layer effect. Therefore the aerodynamic drag coefficient of the bus remained stable after reaching the Reynolds number 57000.In this case, aerodynamic drag coefficients of blunt vehicles such as juggernauts and trucks could be determined with small-scale of these vehicles and low speed wind tunnel via Reynolds number independence.

5. References

1. İçingür, Y., Solmaz, H., “Determination of drag coefficients of various automobile models in a low speed wind tunnel”, Journal of the faculty of engineering and architecture of Gazi University, 2, 60-62, 2011.
2. Hucho, W. H., “Aerodynamics Of Road Vehicles”, Butterworth, London, 106-150, 298-410 (1998).
3. Gürlek, C., “Study of the flow around a bus”, PhD Thesis, Çukurova University Graduate School of Natural and Applied Sciences, Adana, 119-122, 2008.
4. Solmaz, H., İçingür, Y., “Importance and determination of lift coefficient for road vehicles”, Journal of Polytechnic, 13, 3, 203-208, 2010.
5. Milliken, W.F., Milliken, D.L., “Race Car Vehicle Dynamics”,Society of Automotive Engineers International, Warrendale, 83-110, 1995.
6. Sumantran, V., Sovran, G., “Vehicle Aerodynamics”, Society of Automotive Engineers International, Warrendale, 483-531, 1996.
7. Barnard, B.H., “Road Vehicle Aerodynamic Design”, Longman, London, 1-20, 1996.
8. Kavadar, G., “Design and protoyping of a mechatronic system as a drag reduction device for buses”, MSc Thesis, İzmir Institute of Technology, İzmir, 80-86, 2006.
9. Wiedemann J., Ewald, B., Turbulance maniplation to increase effective Reynolds numbers in vehicle aerodynamics. AIAA J. 27, 763-69, 1989.
10. Chuan, F.J., Tao, Y., “Low speed pressurized wind tunnel at home which is building”, Journal of Experiments in Fluid Mechanics, 19, 3, 1-6, 2005.
11. White, M.F., “Akışkanlar Mekaniği”, McGraw Hill, İstanbul, 210-220, 2004.
12. Çengel, Y.A., Cimbala, J.M., “Akışkanlar Mekaniği” McGraw Hill, İzmir, 299-300, 2008.
13. Solmaz, H., “Determination drag coefficient of different vehicle models in a wind tunnel”, MSc Thesis, Gazi University Institute of Science and Technology, Ankara, 10-50, 2010.
14. Houghton, E. L. And Carpenter, P. W., “Aerodynamics For Engineering Students”, Butterwworths, London, 464-465, 2003.

15. Aka, H., “Otomobillerin aerodinamik karakteristiğinin bir rüzgar tünelinde incelenmesi”, MSc Thesis, Gazi University Institute of Science and Technology, Ankara, 67-69, 2003.
16. Gao Feng, Xie Jinfa, Wei Daofu, "Numerical simulation 3-D turbulent flow around the sedan", Chinese Journal of Mechanical Engineering, 36, 6, 48-51, 2000.
17. Gu Zhengqi, Jing Lehua, WU Jun, "The Numerical Analysis and computer simulation on automobile flowfield", Journal of Acta Aerodynamica Sinica, 18, 2, 188-192, 2000.
18. Lanfang Jiang, Hong Liu, Guozhong Chai, Guangnan Jiang, Weiming Lin, “Effect of Automotive Headlamp Modeling on Automotive Aerodynamic Drag”, Proceeding of CAID&CD'2008, Kunming, 588-593, November 2008.