EXPERIMENTAL OPTIMIZATION OF AERODYNAMIC DRAG COEFFICIENT OF A MINIBUS MODEL WITH NON-SMOOTH SURFACE PLATE APPLICATION

CIHAN BAYINDIRLI[1*], MEHMET CELIK[2]

[1]*Nigde Omer Halisdemir Universty, Nigde Vocational School of Technical Sciences, 51100, Nigde, Türkiye*

[2]*Karabuk University, Faculty of Engineering, 78050, Karabuk, Türkiye*

Abstract: In this study non-smooth surface plate application was experimentally investigated to flow control around a scaled minibus model in a wind tunnel. 1/15 scaled minibus model and non-smooth surface plate were designed in SolidWorks® Cad program and produced in 3-D printer. It was focused on decrease of drag coefficient of vehicle with non-smooth surface plate by reducing flow separation. The experimental tests carried out 4 free stream velocities between the speed of 13.90-27.40 m/s and $2.62x10^5$-$5.18x10^5$ Reynolds numbers. In wind tunnel tests Reynolds number independence was used to ensure dynamic similarity. The blockage rate was 10.68 %. It was determined that the using of non-smooth surface plate e on the front roof area of minibus model decreased to drag coefficient by an average of 1.03%. This reduction rate can decrease fuel consumption of vehicle about 0.5% at high speeds.

Keywords: drag coefficient, wind tunnel, CAD, aerodynamic, non-smooth surface plate

1. INTRODUCTION

The flow control and flow separation of road vehicles are major research and development subject of researches and designers in aerodynamic studies. Aerodynamic drag force is directly related to the fuel consumption and the exhaust emissions of vehicles. Therefore, it is very important to examine the flow structures around the vehicles both experimental or numerical methods and to optimize the vehicles in terms of aerodynamics. The thermal efficiency of internal combustion engines approached the theoretical limits. Material technology is the most important factor limiting further improvement of thermal efficiency. Increasing of fuel prices, decreasing oil resources and greenhouse gas emissions force to automotive manufacturers to more efficient aerodynamically designed. The drag force of a road vehicle is responsible for a large part of the vehicle's fuel consumption and contribute up to 50% of the total vehicle fuel consumption at high vehicle speeds. There are many scientific studies on reducing aerodynamic drag of ground vehicles and increasing of their efficiency. These studies focused on decrease of pressure based drag force and air flow separation around of vehicles. These methods entitles as active and passive flow controls methods. According to Belman et.al (2010) the fuel consumption by vehicles accounts for over 30% of CO_2 and other greenhouse gas emissions. Moreover, most of the usable energy from the engine goes into overcoming the aerodynamic drag (53%) and rolling resistance (32%); only 9% is required for auxiliary equipment and 6% is used by the drive-train. 15% aerodynamic reduction at highway speed of 55 mph can result in about 5–7% in fuel saving [1]. Gopal and Senthilkumar (2012) examined on effects of vortex generator application for a passenger car. The variation of pressure coefficient, dynamic pressure, coefficient of lift and drag

[*] Corresponding author, email: cbayindirli@ohu.edu.tr
doi.org/10.29081/jesr.v28i4.004

with and without vortex generators on the roof of a utility vehicle for varying yaw angles. The yaw angles used are 10°, 15° and 20°. To measure the effect of altering the vehicle body, wind tunnel tests have been performed with 1:15 scaled model of the utility vehicle with velocities of 2.42, 3.7, 5.42 and 7.14m/s. The experiments showed that minimum of 20% reduction in drag is obtained for VG with a yaw angle of 10˚ [2]. Song et al. (2012) aerodynamically optimized outer shape of a sedan car by using an Artificial Neural Network (ANN) method. They focused on modifying the rear body shapes of the sedan car. To determine the optimization variables, the unsteady flow field around the sedan driving at very fast speeds was analyzed by CFD simulation, and fluctuations of the drag coefficient ($C_{D)}$ was calculated. When compared to the base model, aerodynamic performance was improved by about 5.64% in aerodynamically optimized shape for the rear end of the sedan [3]. Moussa et. al. (2015) investigated and optimized the effect of multiple bumps placed in the rear end of the cabin roof on the overall aerodynamic drag reduction for a generic model of a commercial truck. The numerical method combined automatic redesign of the add-ons, simulation, and optimization using a globalized form of the Taguchi method. The numerical optimization showed that drag reduction can be achieved at different values of the design parameters with an overall expected reduction between 6 and 10%. Overall, the bumps increased the cabin surface pressure coefficient and displaced the attachment of the bed flow over the tailgate toward the cabin, eventually reducing the size of the recirculating flow behind the tailgate and improving the pressure there [4]. Abinesh and Arunkumar (2014) aimed to modify the outer surface and structure of the bus aerodynamically to reduce the effect of drag force of the vehicle which in turn results in reduction of fuel consumption of the vehicle. Two prototype bus body were modeled by using CFD to reduce the drag force. As a result they increased performance and reduced the fuel consumption. The reduction in aerodynamic drag force was 10% [5]. The flow structure around of base minibus model investigated by Bayindirli and Celik, (2018) in their study [6]. They calculated C_D coefficient of a minibus model as 0.415 in Fluent. Wang et.al. (2017) were carried out numerical simulations to investigate the influence of the dimpled nonsmooth surface on an Ahmed body model. In that numerical simulations were carried out to investigate the influence of the dimpled nonsmooth surface which was added on the rear slope of a general vehicle body on the reduction in the aerodynamic drag. The decrease in the negative zone the turbulence kinetic energy and the vorticity in the wake showed that the dimpled non-smooth surface had a positive effect on the reduction in the drag. The DOE analysis revealed that the D/S ratio is the key parameter for the dimples to reduce the aerodynamic drag and the results showed that the optimal combination of the design variables can reduce the aerodynamic drag coefficient by 5.20% [7]. An Improved Delayed Detached Eddy Simulation (IDDES) was used to determine of the effect of 2 vortex generators (vg) and riblets types on drag force. The Ahmed body model was tested at a 25° slant in IDDES and verified by experimentally. It was determined that these 2 drag-reduction devices significantly affected flow separation from surface. A 6.21% drag reduction was achieved using vg. Finally, 4 combinations of those flow control parts were tested and 8.62% drag reduction was achieved [8]. In another study, excellent hydrodynamic features of tuna were investigated experimentally. Three types of bionic surfaces were designed and manufactured based tuna skin. Its mechanical properties and construction inspired and exemplified in study. The features of surface and coating of bionic surfaces were investigated. The effect of them to drag reduction performance was determined. As seen in result that the drag reduction effect was proportional to the flexible coating thickness. A 7.22% drag reduction was achieved in the dual structure coupling surface [9].

The aim of this study is numerically determine the effect of non-smooth surface plate to aerodynamic drag and fuel consumption on a minibus model. The original part of the study is determination of effect of this passive flow method on the flow structure and reduction in C_D by reducing the flow separation.

2. EXPERIMENTAL SETUP

2.1. Material and method

In this study, non-smooth surface plate was used to control flow around of model vehicle. This method is a passive flow control method because of no energy expedinture from engine of vehicle. The non-smooth surface plane and vehicle model designed by using computer aid design program. The models were producted in 3-D printer. The model minibuses and non-smooth plate are shown in Figures 1- 3. The non-smooth surface plate was placed at the end of the windshield.

In studies on vehicle aerodynamics, geometric, kinematic and dynamic similarity conditions have to provide between prototype and model car. To provide geometric similarity, model vehicle drawn in CAD program and produced in 3-D printer using this drawing data. So, first similarity condition was obtained. In kinematic similarity depends on blockage ratio factor [10, 11]. It was provided in wind tunnel tests. Finally, to provide dynamic similarity Reynolds number independence was used as stated in the literature for wind tunnel experiments. The aerodynamic drag coefficient C_D is the function of the drag force (F_D), density (ρ), free flow velocity (V) and front view area (A) and it was presented equation 1. As a results of the wind tunnel tests, the drag force was measured in different free stream velocities and the C_D coefficients were calculated.

$$C_D = \frac{F_D}{\frac{1}{2}\rho V^2 A} \tag{1}$$

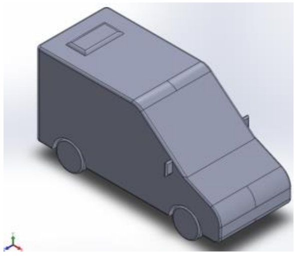

Fig. 1. Drawing data of model minibus (Izometric viev) [6].

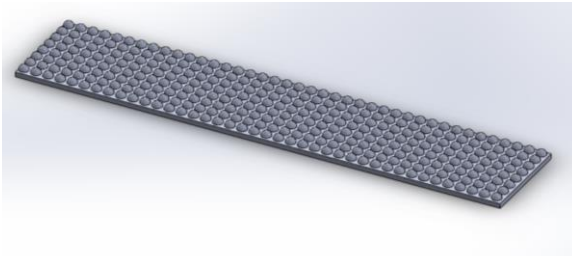

Fig. 2. Drawing data of non-smooth surface plate.

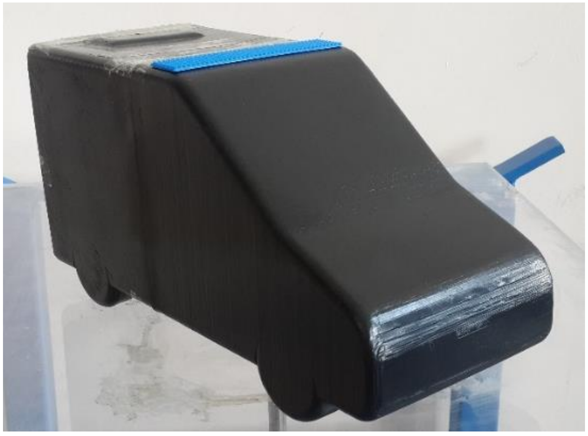

Fig. 3. Test model vehicle with non-smooth surface plate.

2.2. Wind tunnel

The experiments were conducted in a low-speed suction-type wind tunnel. The maximum free stream velocity was 30 m/s in wind tunnel. The sizes of test are 40×40×100 cm. The desired free stream velocity was obtained by 4 kW powered axial fan motor in the test region. A frequency inverter was used to control of rpm of fan motor in the range of 0-50 Hz and has 0.1 Hz step. The experimental tests were carried out in the range of 2.62×10^5 – 5.18×10^5 Reynolds numbers. The wind tunnel and test devices are given in Figure 4.

In experimental studies, a Honeywell Model 41 load cell was used to measure the drag force with 0.1% accuracy. The load cell can measure 0-5 lb force, between the value of 0 - 5 Vdc output voltages. The drag force measurements were conducted on 20 seconds at 1000 Hz frequency for each free stream velocity and the average of these 20000 values were taken to obtain average drag force. The free stream velocities were calculated by using a pitot tube. The 24-bit OROS OR35 real-time multi-analyzer, external recorder with a 40 kHz sampling frequency and OROS Navigate data acquisition software were used to collect voltage outputs from the load cell [12].

Fig. 4. Experimental setup [8].

3. RESULTS AND DISCUSSION

3.1. Drag coefficient of model minibus

As seen in Table 1 and Figure 5, C_D coefficient of the minibus model was determined as 0.482 at 4 different Reynolds number. It was numerically determined as 0.415 by Bayindirli and Çelik (2018). This CFD result supports wind tunnel result. The difference between the results may be due to the uncertainty in the experimental setup or the quality of the mesh structure in the numerical analysis.

Table 1. C_D coefficient of model minibus in wind tunnel.

Velocity (m/s)	Reynolds Number	Base Model C_D
13.90	262907	0.476
17.95	339411	0.485
22.54	426218	0.487
27.40	518174	0.481
	Average	**0.482**

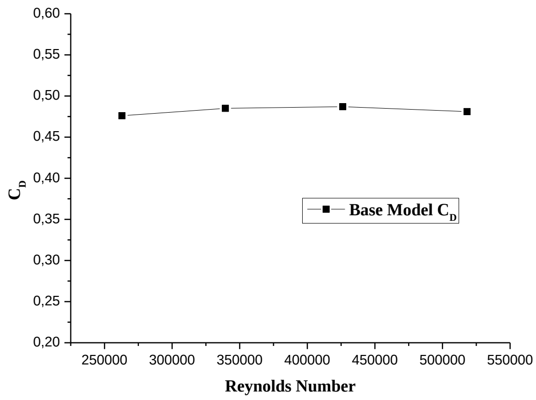

Fig. 5. Aerodynamic drag coefficient (C_D) graph of base model minibus.

3.2. Drag coefficient of optimized minibus

As seen in Table 2 and Figure 6, C_D coefficient of the optimized minibus model which has non-smooth surface plate was determined as 0.477. When compared with base minibus model there was 1.03% drag reduction average. CFD analysis of the vehicle model and non-smooth surface plate were carried out by Bayindirli and Celik (2019). In the related study, 2.61% improvement was achieved. The numerical result and experimental result reveal that lower drag force can be obtaien by decreasing flow seperation with non-smooth surface plate.

Table 2. The C_D coefficient of optimized model bus and reduction rates.

Velocity (m/s)	Reynolds Number	Base Model C_D	Optimized model C_D	Drag Reduction
13.90	262907	0.476	0.472	0.85%
17.95	339411	0.485	0.479	1.25%
22.54	426218	0.487	0.479	1.60%
27.40	518174	0.481	0.479	0.45%
	Average	**0.482**	**0.477**	**1.03%**

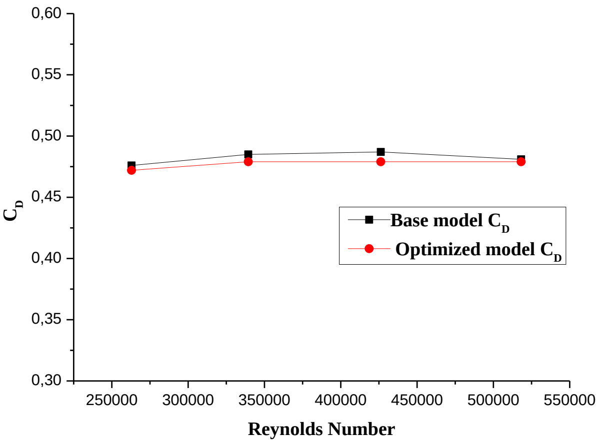

Fig. 6. Aerodynamic drag coefficient (C_D) reduction graph of optimized minibus.

Flow imaging around of base minibus model were given in Figure 7. As seen in the images, there is a flow separation depending on the angle of attack of windshield and geometry of front roof area. This non-smooth plate chaneges to flow turbulent. It was aimed to hold the flow on surface. This is the source of the achieved improvement.

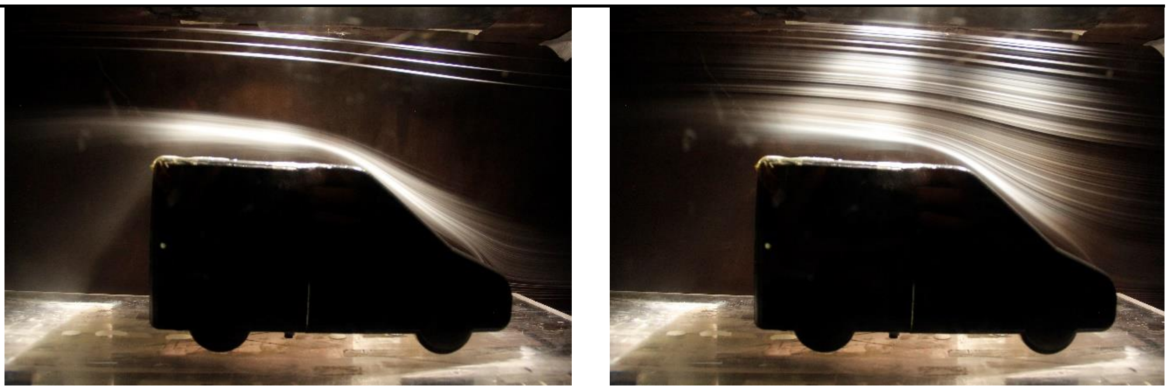

Fig. 7. Flow visualizations of base minibus model.

In this study, wind tunnel tests were conducted to determine the effect of the nonsmooth surface plate to drag force. Thereby flow separation was delayed by this flow control method application and drag reduction obtained. The decrease in the negative zone and the vorticity in the wake area showed that non-smooth surface had a positive effect on the drag force. Flow visualizations around optimized minibus model were obtained in Ansys Fluent program and they were given in Figure 8 and 9. Ansys Fluent is an analysis program that is used for product or system analysis in many engineering applications and contains many modules in itself.

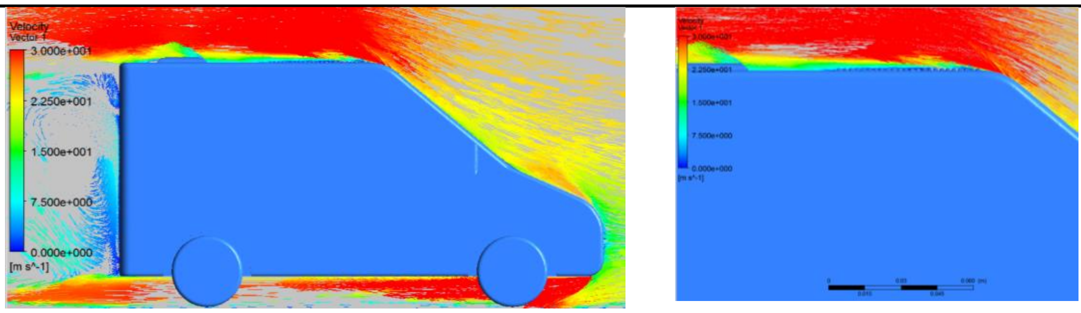

Fig. 8. The vector image of optimized model vehicle at 5.4x10^5 Reynolds [11].

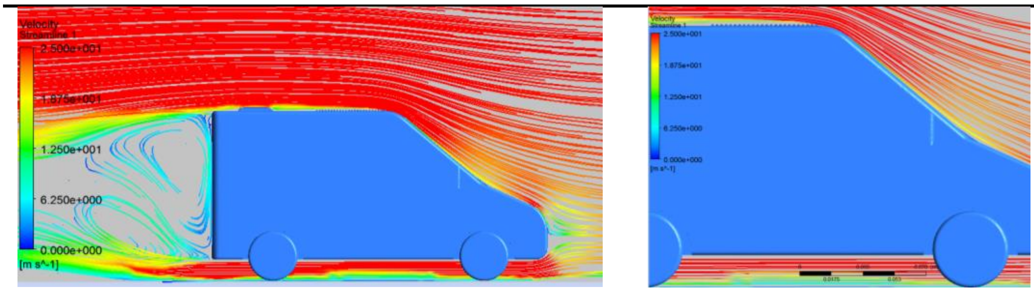

Fig. 9. The streamline image of optimized model vehicle at 5.4x10^5 Reynolds [11].

4. CONCLUSIONS

In this study drag coefficient of a minibus model was experimentally determined in four different Reynols Number. The C_D coefficient was calculated as 482. This result coherent with literature data. To decrease drag coefficient a non-smooth plate mounted on front of area of base minibus model. It was aimed to decrease or delay flow separation by using passive flow control. When flow separation decreased negative pressure area also decreased. Non-smooth surface creates drag force but its pozitive effect to flow separation and negative pressure area compensated for this negativity. So, drag coefficient was reduced 1.03%. It Its effect to fuel consumption 0.5% about at high vehicle speeds.

NOMENCLATURE

A	Frontal area of bus model, m^2
C_D	Drag coefficient
F_D	Drag force, N
u_∞	Free stream velocity, m/s
Re	Reynolds number
ν	kinematic viscosity, m^2/s
CFD	Computational Fluid Dynamics
Exp.	Experimental
Afc	Active flow control
Pfc	Passive flow control

REFERENCES

[1] Bellman, M., Agarwal, R., Naber, J., Chusak, L., Reducing energy consumption of ground vehicles by active flow control, ASME 2010 4th International Conference on Energy Sustainability, American Society of Mechanical Engineers, 2014, p. 785-793.

[2] Gopal, P., Senthilkumar, T., Aerodynamic drag reduction in a passenger vehicle using vortex generator with varying yaw angles, ARPN Journal of Engineering and Applied Sciences, vol. 7, no. 9, 2012, p. 1180-1184.
[3] Song, K.S., Kang, S.O., Jun, S.O., Park, H.I., Kee, J.D., Kim, K.H., Lee, D.H., Aerodynamic design optimization of rear body shapes of a sedan for drag reduction, International Journal of Automotive Technology, vol. 13, no. 6, 2012, p. 905-914.
[4] Moussa, A.A., Fischer, J., Yadav, R., Aerodynamic drag reduction for a generic truck using geometrically optimized rear cabin bumps, Journal of Engineering, vol. 2015, 2015, p. 2-14.
[5] Abinesh, J., Arunkumar, J., CFD analysis of aerodynamic drag reduction and improvement fuel economy, International Journal Mechanical Engineering and Robotics Resarch, vol. 3- 4, 2014, p. 430-440.
[6] Bayındırlı, C., Çelik, M., The investigation of flow structure around of a minibus model by CFD method, IV International Academic Resarch Congress, Alanya, Türkiye, 2018.
[7] Wang, Y., Wu, C., Tan, G., Deng, Y., Reduction in the aerodynamic drag around a generic vehicle by using a non-smooth surface, Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering, vol. 231, no. 1, 2017, p. 130–144.
[8] Yang, X., Hu, Y., Gong, Z., Jian, J., Liu, Z., Numerical study of combined drag reduction bases on vortex generators and riblets for the ahmed body using IDDES methodology, Journal of Applied Fluid Mechanics, vol. 15, no. 1, 2022, p. 193-207.
[9] Chen, D., Chen, H., Cui, X., Dual-coupling drag reduction inspired by tuna skin: fan-shaped imbricated fish scale composited with flexible coating, AIP Advances, vol. 12, no. 035218, 2022.
[10] Çengel, Y.A., Cimbala, J.M., Fundamentals of fluid mechanics and applications, Ed. McGraw-Hill Companies, 2008, p. 562-599.
[11] Bayindirli, C., Celik, M., The numerical determination of non-smooth surface to drag coefficient on a minibus model, 5th International Conference on Sustainable Development (ICSD), Belgrad, Serbia April 17-19, 2019.
[12] Cihan, B., Akansu, Y.E., Mehmet, C., Experimental and numerical studies on improvement of drag force of a bus model using different spoiler models, International Journal of Heavy Vehicle Systems, vol. 27, no. 6, 2020, p. 743-776.