INTERNATIONAL JOURNAL OF AUTOMOTIVE SCIENCE AND TECHNOLOGY

2021, VOL. 5, NO: 4, 412-418 www.ijastech.org e-ISSN: 2587-0963

Reducing of Pressure Based Drag Force of a Bus Model by Flow Control Rod in Wind Tunnel

Cihan Bayindirli[1*]

0000-0001-9199-9670

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

Abstract

The aerodynamic force of vehicles has been affected by the geometric shape of front surface. The aerodynamic force of a scaled bus model and C_D were experimentally determined in wind tunnel. The experimental tests were conducted at 6 different free flow velocities. They were performed in the range of 3.8×103-7.9×103 Reynolds number. The drawing data of model bus were obtained by 3-D scanner and drawn in SolidWorks® program. The circular cross section flow control rod was designed in 10 mm, 20 mm and 30 mm diameter and positioned at 3 different locations (L/H) on the model bus. By this passive flow control method, aerodynamic drag reductions were obtained up to 10.06%, 7.35% and 7.85% respectively.

Keywords: Aerodynamic, Bus model, Drag coefficient, Passive flow control, Wind tunnel

Research Article

https://doi.org/10.30939/ijastech..994351

Received 12.09.2021
Revised 01.11.2021
Accepted 13.12.2021

* Corresponding author

Cihan Bayindirli

cbayindirli@ohu.edu.tr

Address: Nigde Vocational School of Technical Sciences, Nigde Omer Halisdemir University, Nigde, Türkiye

Tel:+903122028653

1. Introduction

In earlier times automotive manufacturers were designed vehicles for high speed. The aerodynamic forces and fuel consumption of vehicles were not priority. Then, they and the researchers were focused to aerodynamically design of cars in order to reduce the fuel consumption and exhaust emissions. Today vehicle manufacturers attach great importance to vehicle performance and research and development to decrease C_D and improve power of engines. they are also searching to reduce the aerodynamic loss of the vehicle during movement. The fuel consumption of vehicles was effected from aerodynamic drag force and it can be minimized by design. To improve drag force and reduce fuel consumption controlling of flow separation necessary around of vehicles. The flow separation can delay or prevent by flow control. They can be classified as passive and active flow control methods. [1]. The passive control technique is modification of vehicles. It has significant advantages when compared active control technique due to consume no energy from the engine power [2]. Examples of active flow control are, synthetic jet, steady blowing, suction, plasma actuator, movable parts and it consume energy from the vehicle. If the C_D coefficient of a vehicles minimized 2% fuel consumption can be reduced about 1% at high speeds [3-6]. Between the rates of 9-17% drag reduction were obtained with better spoiler design, and 4-6% by trailer skirts and 0-4% by closing air gaps. By reducing C_D fuel consumption can be improved between the rate of 1-17% on truck trailer model [7]. The drag force of sedan racing car was reduced by up to 22.13% with 5 buffer diffuser as passive flow technique. By using the CFD method three different bus models were investigated to reduce the drag force of the bus model. The front and rear surface of the bus were modified [8]. By improving of rear bodywork can decrease C_D coefficient up to 26% [9]. The C_D of a heavy vehicle was reduced by some pfc parts such as spoiler and closing the distance between truck trailer combination. The effect of application of synthetic air jet on Ahmed body model was determined by CFD method. The C_D was decreased by 3-9% with continues air injection rear of vehicle model [10]. In another active flow control study C_D was reduced drag force 3% by delaying flow separation on vehicle model [11].

Buses have been used very extensively in passenger transportation. They have been driving between intercity at high speeds. So aerodynamic force of the busses significantly effects to fuel consumption. The aim of this study is reduce pressure based drag force of a bus model by pfc method. Drag reductions were experimentally obtained between the percent of 7.35% - 10.06% by flow control rod application. The original part of this study is that the aerodynamic improvement with the flow control rod is capable of to be product and applicable in real model.

2. Material and Methods

Except of the moving ground three necessary similarity conditions were provided in studies both experimental and numerical. A licensed model bus was used to provide geometric similarity, the. For kinematic similarity blockage rate was 6.81%. It was recommended that blocking rate should be lower than 7.5% in order to provide kinematic similarity [12-13].

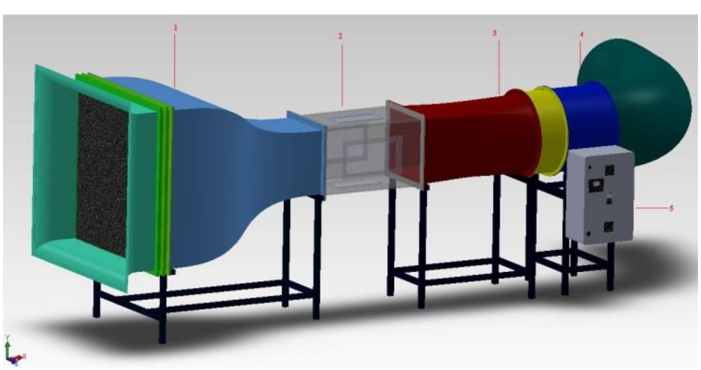
The Re is defined as the ratio of inertial forces to viscous forces as seen Eq.1.

$$R_e = \frac{U_\infty L}{\nu} \qquad (1)$$

Re has to be same for model and prototype car in aerodynamic studies. But the size of prototype and model vehicle so different. So it is very difficult to achieve same Re in a wind tunnel. So Re independency uses to providing dynamic similarity. In this study Re independency was used to provide dynamic similarity.

2.1. Experimental Setup

The tests were made in an absorption kind wind tunnel. The highest flow speed was 30 m/s and test area size is 400×400×1000 mm. The desired flow speed was provided by fan motor provided was used to obtain in the test region. The control of rpm was controlled by frequency inverter. It was 0.1 Hz step and range of 0-50 Hz. The tests were conducted between 3.8×10^5–7.9×10^5 Re. Experimental setup was given in Fig. 1 (a, b).



1.Contraction cone 2. Test section 3. Diffuser 4. Fan unit 5. Frequency inverter
Fig. 1(a). Experimental setup [5]

Fig.1(b). Experimental setup [5]

Drag forces were measured by load cell with 0.1% accuracy in experimental studies. It can calculate force to 5 lb and to 5 V voltage. The aerodynamic force measurements were made on 20 seconds. The sample was 1000 Hz for each test. To calculate average force 20000 values were used for every test. A pitot tube was used to determine flow speed in test section. A real time multi analyzer and external recorder were used in studies. To read to voltage outputs data acquisition software were used. Related test devices are seen in Fig.2-3.

Fig. 2. Honeywell 41 Model load cell

Fig. 3. OROS OR35 Multichannel data acquisition system

The experiments were conducted on the test model. It was given in Figure 3-4. The bus model is 1/33 scaled model of real size bus. The measurements of model bus were precisely obtained by 3-D scanning method. The drawing data of bus and flow control rod designed by SolidWorks program. Then they were produced in 3-D printer. The sizes of bus model are 101×96.3×44 mm [14]. The produced models and application on bus model were given in Fig.4-6.

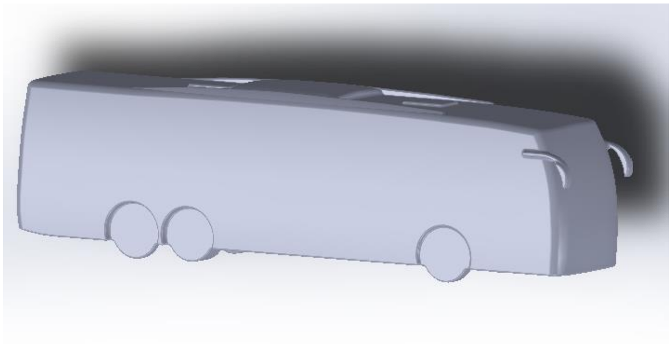


Fig. 4. The bus model in Solid Works program [14].

Fig. 5. Produced bus model [14].

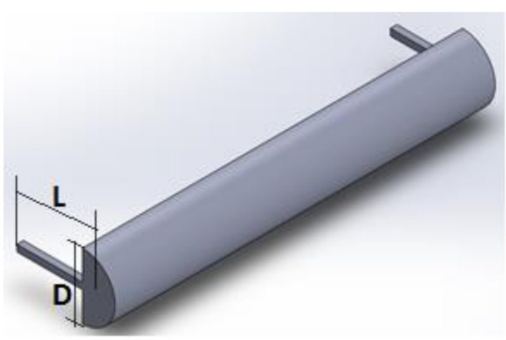


Fig. 6. Solid Works drawing of a circular section flow control rod (D=10mm)

It is thought that the flow control rod mechanism can be control by a hydraulic arm or with an electrical motor and stay under the front bumper when the bus within the city. It opens in the intercity roads by driver controlled. The mechanism of the flow control rod is given in Fig. 7. The aims of this study determine of this flow control road to drag force which is pressure based on front of the bus.

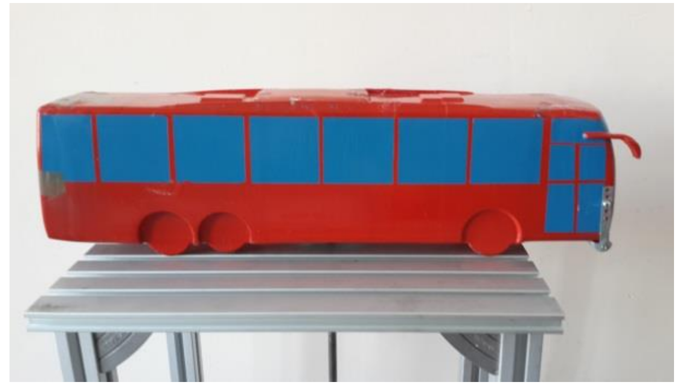


Fig. 7a. The operation of circular section flow control rod mechanism

Fig. 7b. The operation of circular section flow control rod mechanism

Fig. 7c. The operation of circular section flow control rod mechanism

2.2. Load cell calibration

The load cell was calibrated to measure correct drag force or take precautions against possible incorrect measurement. The related voltages were read by data acquasion system for 9 different weights on the load cell. As seen in Table 1, the voltages were obtained according to corresponding weights. The calibration equation was obtained and linear fitting graph was drawn in Fig.8 [5].

Table 1. Load cell calibration values

Weight (Kg)	Voltage (V)	Force (N)
0.368	3.04	3.45
0.324	2.85	3.17
0.284	2.67	2.91
0.223	2.20	2.21
0.172	1.85	1.70
0.122	1.54	1.25
0.071	1.20	0.74
0.021	0.83	0.19
0	0.67	0

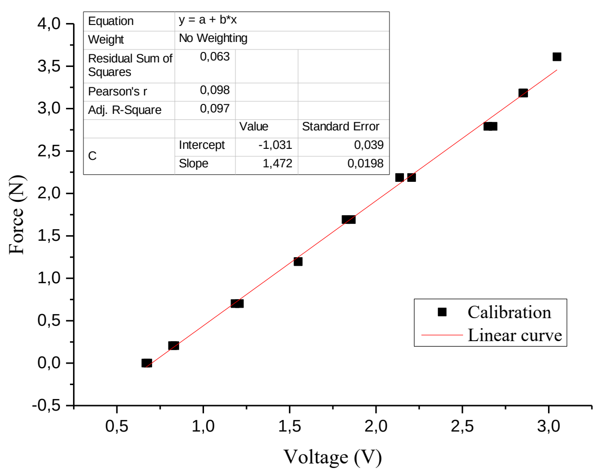


Fig. 8. Calibration graph

2.3. Uncertainty Analyses

The uncertainties of experimental setup determine accuracy of experimental results. The uncertainty values of the obtained parameters were given in Table 2.

Table 2. Uncertainty values of calculated parameters [5]

Calculated Parameter	Uncertainty value (%)
Reynolds Number	3.87
Drag force	4.5
Drag coefficient	4.7

3. Experimental Results

3.1. The C_D of base bus model

The C_D value of base bus model was experimentally determined as 0.633 averages in study of by Bayindirli and Celik, (2018). This article focuses on reducing of this C_D value. This C_D value coherent with real bus C_D in the literature.

3.2. The C_D of model 1 (D=10mm)

The C_D of the model 1 was respectively calculated as 0.570. 0.620 and 0.614 as given in Table 4 and Fig 10. The minimum C_D value was calculated in L/H=0.10. The improvement rate on C_D coefficient is %10.06.

Table 4. C_D values of model 1 (D=10mm)

Re	Base C_D	L/H= 0.10	L/H= 0.15	L/H= 0.20
3.8×10^3	0.583	0.521	0.580	0.599
4.6×10^3	0.630	0.543	0.608	0.625
5.4×10^3	0.629	0.568	0.642	0.619
6.2×10^3	0.654	0.587	0.643	0.628
7.1×10^3	0.653	0.600	0.621	0.603
7.9×10^3	0.651	0.598	0.627	0.612
Average	**0.633**	**0.570**	**0.620**	**0.614**

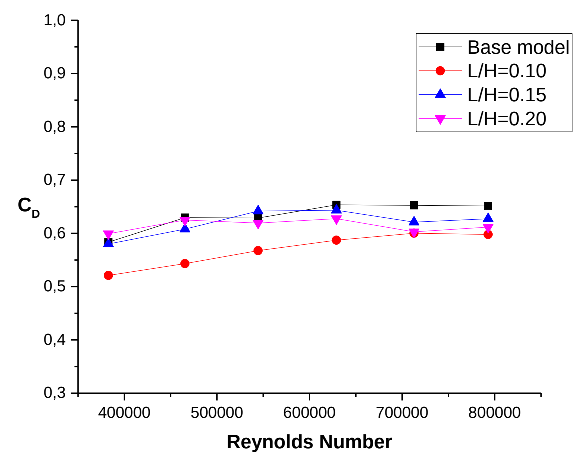


Fig. 10. The C_D values of model 1

3.2. The C_D of model 2 (D=20mm)

The C_D values of the model 2 were obtained as 0.587. 0.640 and 0.623. The minimum C_D was calculated in L/H=0.1 as 0.587. As seen in Table 5 and Fig.11 the maximum drag reduction rate is 7.35%.

Table 5. Aerodynamic C_D values of base model 2 (D=20mm)

Re	Base C_D	L/H= 0.10	L/H= 0.15	L/H= 0.20
3.8×10^3	0.583	0.651	0.636	0.556
4.6×10^3	0.630	0.610	0.627	0.609
5.4×10^3	0.629	0.583	0.642	0.649
6.2×10^3	0.654	0.528	0.637	0.657
7.1×10^3	0.653	0.561	0.636	0.641
7.9×10^3	0.651	0.588	0.660	0.627
Average	**0.633**	**0.587**	**0.640**	**0.623**

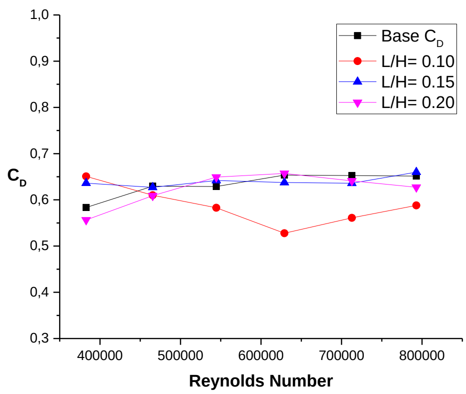


Fig. 11. The C_D values of model 2

3.3. The C_D of model 3 (D=30mm)

As given in Table 6 and Fig. 12, the model 3 C_D values were respectively calculated as 0.592. 0.584 and 0.720. The minimum C_D was 0.584 in L/H= 0.15. Maximum drag reduction was 7.85%.

Table 6. Aerodynamic C_D values of base model 3 (D=30mm)

Re	Base C_D	L/H=0.10	L/H=0.15	L/H=0.20
3.8×10^3	0.583	0.515	0.478	0.763
4.6×10^3	0.630	0.574	0.575	0.655
5.4×10^3	0.629	0.598	0.602	0.734
6.2×10^3	0.654	0.617	0.631	0.735
7.1×10^3	0.653	0.626	0.638	0.744
7.9×10^3	0.651	0.623	0.578	0.688
Average	**0.633**	**0.592**	**0.584**	**0.720**

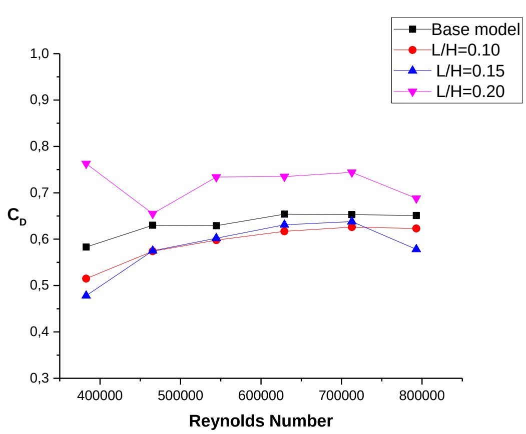


Fig. 12. The C_D values of model 3

Two kinds of flow control techniques were used to control air flow around of solid parts in aerodynamic. They were categorized as active flow control and pfc based on energy expenditure. The pfc is a flow control method which is realized by the changing form of geometry which is examined aerodynamically without spending any energy. Many flow control methods were discussed in literature in related papers. Some methods were applied individually and some of them were combined. Every control method can reduce aerodynamic drag reduction. It was expressed that the C_D reduction could be as much as 20% by using the pulse jet [16, 17] and plasma actuator [18] as active control technique. Besides the drag could be decreased to 21.2% by using flaps [19]. The drag reduction can reduce as much as 30% by combined flow control technique like as blowing jets with porous layer [20]. It is necessary to control of flow around objects in order to lower drag force. Their attachments are little bit easier than pfc. Main disadvantage of active flow control is that it requires power from engine and application of this method increase fuel consumption of vehicles [21]. 5.03% drag minimization was obtained in another study using the afc-pfc techniques for an air vehicle [22]. The efficiency of passive-active flow technique was investigated to achieve lower aerodynamic force. That paper mainly focuses on the methods employed to prevent or delay air flow separation by using vortex generator (VG), spoiler, and splitter pfc methods been reviewed. 50% fuel consumption of road vehicles result from drag force. [23]. These literature values support result of this paper.

In this study a pfc method was used to enhance the flow structure around the model bus and to bring it out of ordinary flow. To obtain drag reduction 3 different diameter circular cross-sectional flow control rod positioned at 3 different distances to on front of model bus. Pfc technique needs changing of shape on the vehicle body. It affects the flow structure thus desirable reduction can obtain. But this method is little bit complex. The total aerodynamic drag consists of friction and pressure induced. In road vehicles a great proportion of total drag force result from pressure induced. The pressure based drag force affects to geometry's surface perpendicularly. The frictional drag force forms from shear stress and it affects the object's surface parallels. In this study the desired drag reduction was obtained by the circular cross section flow control bar. The high pressure area was decreased on front of bus model and desired flow direction was obtained. It was shown in flow visualizations that the air flow on the front bumper and windshield surface was transferred to the windshield surface by the control rod. Thus the pressure based aerodynamic was reduced by leaving the front surface region of the bus in the negative pressure area in the turbulent region.

4. Conclusions

Fuel consumption is reduced by 1% when the aerodynamic C_D values of vehicles are reduced by 2% at high speeds [24]. Fuel consumption of these vehicles a significant concern for consumers. Because a significant part of the transportation cost is related to fuel consumption. In this paper, aerodynamic force and flow structure around bus model were investigated. The positive effect of pfc method on drag force is revealed. The base bus C_D was calculated as 0.633 in wind tunnel. 3 different diameters circular cross-sectional flow control rod was installed at three different distances to bus. The maximum experimentally drag reduction was 10.06%. This reduction rate can be decreased fuel consumption by about 5% at high speeds (over96 km/h). The total drag force forms from %89.07 pressure induced and 10.93% friction induced. It is determined that there is a big potential to decrease friction induced drag force for buses. It was experimentally determined that the circular cross-sectional flow control rod application improved the flow structure around of bus.

Acknowledgment

This study was supported by Research Projects Coordination Unit of Nigde Omer Halisdemir University with ref. number FEB 2016/24 BAGEP.

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
ρ	: Density of air, kg/m^3
H	: Height of model bus, mm
L	: Distance between model bus-flow control rod, mm
D	: Diameter of flow control rod, mm
V	: Speed of vehicle km/h
LES	: Large Eddy Simulation
RNG	: Renormalization-Group
Exp	: Experimental
AFC	: Active flow control
PFC	: Passive flow control

Conflict of Interest Statement

The author declares that there is no conflict of interest in the study.

References

[1] Cattafesta L N, and Sheplak M. Actuators for active flow control. Annual Review of Fluid Mechanics. 2011; 43(1). 247–272.

[2] Altaf A, Omar A, Asrar W. Review of passive drag reduction techniques for bluff road vehicles. IIUM Engineering Journal.2014; 15(1). 61-69.

[3] Wood R M, and Bauer S. Simple and low cost aerodynamic drag reduction devices for tractor-trailer Trucks. SAE Technical Paper.2003; 01–3377. 1-18.

[4] Cui W, Zhu H, Xia C, Yanga Z. Comparison of steady blowing and synthetic jets for aerodynamic drag reduction of a simplified vehicle. Procedia Engineering.2015; 126. 388 – 392.

[5] Bayındırlı C, Akansu Y E, Celik M. Experimental And Numerical Studies On Improvement Of Drag Force Of A Bus Model Using Different Spoiler Models. Int. J. Heavy Vehicle Systems. 2020; 27-6 743-776.

[6] Hucho W H, and Sovran G. Aerodynamics of road vehicles. Annual Review of Fluid Mechanics. 1993; 25(1). 485-537.

[7] Mohamed-Kassim Z, and Filippone A. Fuel savings on a heavy vehicle via aerodynamic drag reduction. Transportation Research Part D. 2010; 15. 275–284.

[8] Hassan S M R, Islam T, Ali M, Islam. Md Q. Numerical Study on Aerodynamic Drag Reduction of Racing Cars. Procedia Engineering. 2014; 90. 308 – 313.

[9] Rohatgi U S. Methods of Reducing Vehicle Aerodynamic Drag. ASME 2012 Heat Transfer Conference Puesto Rico.USA.

[10] Cui W, Zhu H, Xia C, Yanga Z. Comparison of steady blowing and synthetic jets for aerodynamic drag reduction of a simplified vehicle. Procedia Engineering. 2015; 126. 388 – 392.

[11] Jonathan M, Erik , Gregory R, Rajan K, Kunihiko T, Farrukh A, Yoshihiro Y, and Kei M. Drag reduction on a flatback ground vehicle with active flow control. Journal of Wind Engineering and Industrial Aerodynamics.2015; 145. 292–303.

[12] Çengel Y A, and Cimbala J.M. Akışkanlar Mekaniği Temelleri ve Uygulamaları. Güven Bilimsel Yayınları. 2008; 562-599.

[13] Özel M, Aygün E, Akansu Y E, Bayindirli C, and Seyhan M. The passive flow control around a truck-trailer model. International Journal of Automotive Engineering and Technologies. 2015; 4 -4. 185 – 192.

[14] Bayindirli C, and Celik M. The experimentally and numerically determination of the drag coefficient of a bus model. International Journal of Automotive Engineering and Technologies. 2018; 7-3: 117-123.;

[15] Bansal R, and Sharma R B. Drag reduction of passenger car using add-on devices, Journal of Aerodynamics. 2014; 1-13.

[16] Bideaux E, Bobillier P, Fournier E, Gilliéron P, El Hajem M, Champagne J Y, Gilotte P, Kourta A. Drag reduction by pulsed jets on strongly unstructured wake: towards the square back control. International. Journal of Aerodynamics. 2011; 1(3/4), 282-298.

[17] Gillieron P, and Kourta A. Massive separation control analysis of the pulsed jet actuators effects. Mechanics & Industry. 2013; 14, 441–445.

[18] Khalighi B, Ho J, Coone J, Neiswander B, Corke T, Han T. Aerodynamic Drag Reduction Investigation for a Simplified Road Vehicle Using Plasma Flow Control. Proceedings of the ASME. 2016 Fluids Engineering Division Summer Meeting.

[19] Tian J, Zhang Y, Zhu H, Xiao H. Aerodynamic drag reduction and flow control of Ahmed body with flaps. Advances in Mechanical Engineering.2017; 9(7), 1-17.
[20] Bruneau C H, Creusé E, Depeyras D, Gilliéron P, Mor tazavi I. Coupling active and passive techniques to control the flow past the square back Ahmed body. Computers & Fluids. 2010; 39,1875–1892.
[21] Mominul I M, and Mohammad Z A. Review on Aerody namic Drag Reduction of Vehicles. International Journal of Engineering Materials and Manufacture.2019; 4(1) 1-14.
[22] Yağız B, Kandil O, Pehlivanoglu, V Y. Drag minimization using active and passive flow control technique. Aerospace Science and Technology, 2012; 17-1,21–31.
[23] Sudin M N, Abdullah MA, Shamsuddin S A, Ramli FR, Tahir M M. Review of research on aerodynamic drag reduction method. International Journal of Mechanical and Mechatronics Engineering IJMME-IJENS. 2014; 14-2, 35–45.
[25] Mohamed E A, Radhwi M N, Abdel Gawad A F. Computional investigation of aerodynamic characteristics and drag reduction of a bus model. American Journal of Aerospace Engineering. 2015; 2 (1-1), 64–73.