Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879

# Conceptual Design and Simulation of Rear Pressure Different Air Flow Correction Device Using Computational Fluid Dynamic (CFD)

Open Access

Muhammad Nur Othman[1,*], Qamar Fairuz Zahmani[1], Mohd Faruq Abdul Latif[1], Muhammad Faridhwan Shofri[1], Ahmad Yusuf Ismail[2]

[1] Faculty of Mechanical and Manufacturing Engineering Technology, Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia
[2] Department of Mechanical Engineering, Institut Teknologi Adhi Tama Surabaya, Sukolilo, Kota SBY, Jawa Timur 60117, Indonesia

**ARTICLE INFO**

***Article history:***
Received 16 April 2019
Received in revised form 21 August 2019
Accepted 2 September 2019
Available online 29 September 2019

**ABSTRACT**

Drag reduction is one of the most significant issues within the automotive industry. Pressure drag contributed more than 80% of the total drag and it is highly dependent on vehicle geometry due to boundary layer separation and formation of wake region behind the vehicle. The drag caused by pressure difference at the rear and front of the vehicle is a major issue for all car segments especially bluff body car segments because it contributed largest fuel consumption. Air suction or blown at the rear of the bluff body can alter the flow pattern of the air hence. This resulted in shrinking at the wake region which resulting in an increase of rear pressure and reduces the pressure difference between front and rear of the body. The objective of this study is designing a concept of drag reduction system that can change the pressure distribution and reduce drag coefficient (CD) of bluff body which is represented as Ahmed body model. The design and testing of the airflow correction device were aided by Computer-aided Design (CAD) and computational fluid dynamic (CFD). The concept of the rear air flow correction device altered the flow pattern which changes the pressure distribution and reduces drag coefficient by 4.5352%. Hence, with full of positive expectation, hopefully, this method of reducing drag by using rear suction or blown air flow correction device will be broadly used for bluff body car segments.

***Keywords:***
Drag reduction; Aerodynamics; Weight region; Boundary layer separation; Turbulence model



## 1. Introduction

High pressure drag was created at the back of vehicle especially vehicle with a bluff body car segment which include multi-purpose vehicle (MPV), van, suburban utility vehicle (SUV), hatchback, and station wagon. This drag caused by large pressure difference between front and rear of the vehicle and as a result increase the fuel consumption [1-3]. A lot of studies done by other researchers

* *Corresponding author.*
*E-mail address: muhammadnur@utem.edu.my (Muhammad Nur Othman)*

to reduce the drag such as study of rear spoiler by Sapienza [4] and Sudin [5], study of bumper by Håkansson [6] and Huminic [7] and study of diffuser by Sudin [5].

Currently the study is focusing on research focused on under rear body modification and exhaust gas redirection hence reducing the pressure difference between rear and front of a vehicle [8]. Still, there is no study that have been done in reducing rear suction of air flow which would further improvise the pressure difference between front and rear of a car.

The main objective of this research was to introduce new rear suction air flow correction device. This device will increase the rear pressure and as a result the pressure differential will also reduce accordingly. Theoretically the drag coefficient reduces by having such a device. It is highly potential to be widely used by bluff body car segments.

The effect of venturi will be utilized in inducing car rear pressure. This simulation is conducted using computational fluid dynamic (CFD) method and Computer Aided Design (CAD). To induce the pressure at the rear of the bluff body by rear suction air flow correction device, the effect of venturi will be utilized. The study will be simulated by using computational fluid dynamic (CFD) method and design with Computer-aided Design (CAD) software.

## 2. Methodology

The body that was generated for this study is Ahmed's body as specified in the introduction. Ahmed's body was drawn by using CATIA V5R21 by Dassault System, the dimension shown as in Figure 1. The dimensions used is the standard Ahmed's body dimension with no slant angle which is broadly used by several researcher such as Ahmed [9], Franck [10] and Johnson [11]. In the modelling, the airflow correction device will be added to the standard Ahmed body.

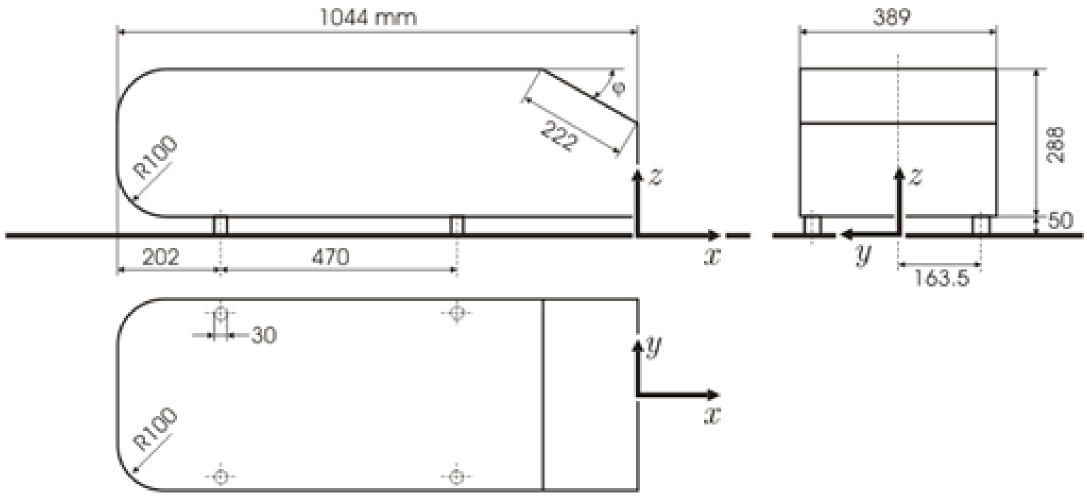


**Fig. 1.** Ahmed body benchmark design

By using simulation, a wind tunnel setup was in accordance to the MIRA model as recommended by Gmbh [12] which is two body length to the front, two body length to the side, two body length to the top and five body length to the rear. After that, Boolean subtract was created to diminish the Ahmed body leaving only air with empty Ahmed body shape inside. The refinement zone was created as suggested by Gmbh [12] and Chen [13]. It is one body length to the front, one body length to the rear, one body length to the top and one body length to the side. The geometry was freeze again. The suggested model was created by combining the suction and blowing effect which applied venturi effect proposed by Bernoulli. The opening at the top rear of the model is ¼ smaller than its bottom rear opening. The small opening yield high-velocity low pressure whereas the large opening yield low-velocity high pressure. Therefore, the air was sucked from top rear opening to the bottom rear opening. The suction at the top rear of the body pull the pass by air and reduce the wake region behind the vehicle. The bottom blowing air increases the air pressure in the wake region.

## 3. Result and Discussion

Referring to Table 1, the suggested model shows a promising lower Drag coefficient (Cd) is 0.2547 with 4.535% reduction compared to the benchmark model. Figure 2 represented a benchmark pressure contour consecutively in the middle of the body longitudinally. Huge wake structures behind the body can be observed from the figure. These unsteady wake structures symbolize large energy losses in the flow which contributes to drag [6]. The wake was generated just behind the body as air separates from the surface at the trailing edge at the rear of the body. A low-pressure area is created behind the body where the separation occurs, and the air is sucked into the wake from all the sides.

**Table 1**
Converged drag coefficient (Cd)

| Models | Converged Drag coefficient (Cd) | % of Reduction |
|---|---|---|
| Benchmark | 0.2668 | 0 |
| Suggested model | 0.2547 | 4.5352 |

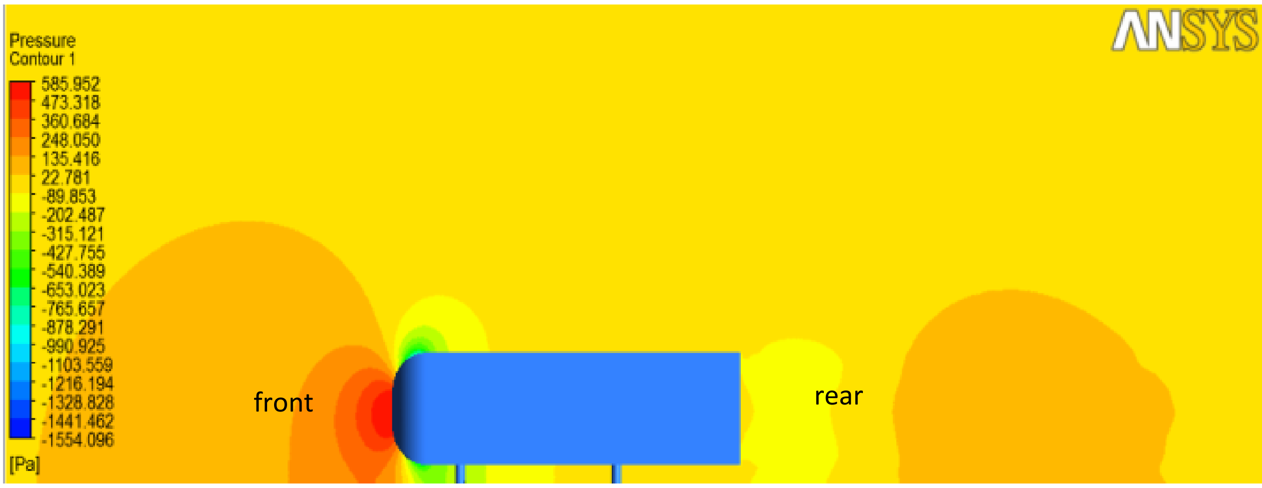

**Fig. 2.** Pressure contour of the benchmark model

Moreover, by observing the graph in Figure 3 it can be clearly seen that there is a large pressure difference between the front of the body and rear of the body which can be further reduced by adding device at the rear of the body. The pressure difference between rear and front of the vehicle contributed to pressure drag which contributes to 80% of overall drag [5]. Based on Figure 4, the pressure in the central of the rear car body is higher compared to the top area and the bottom area. The graph was plot based on the vertical pressure at the rear car body. The graph shows a relatively symmetrical shape from middle area to upward and from middle area to downward. The only difference in shape is at the both top area and bottom area last end. To overcome this, the suggested model was design so that is has a small opening at the top area and large opening for the bottom area. By having such design at the car's rear, a venturi effect would be induced. The outcome would be a high pressure for larger opening and vice versa [14].

Venturi effect thus create significate reduction of pressure different. This is clearly shown in Figure 3, where the wake region become smaller compare to benchmark model. By increasing the rear pressure, the pressure difference between front and rear is reduce and further reduce the drag by 4.5352%. The air at the top rear was suck by the venturi effect and it was then blow at the rear bottom, thus reduce wake region and it also reduce energy lost to overcome friction due to wake and pressure difference.


**Fig. 3.** Pressure contour of the suggested model

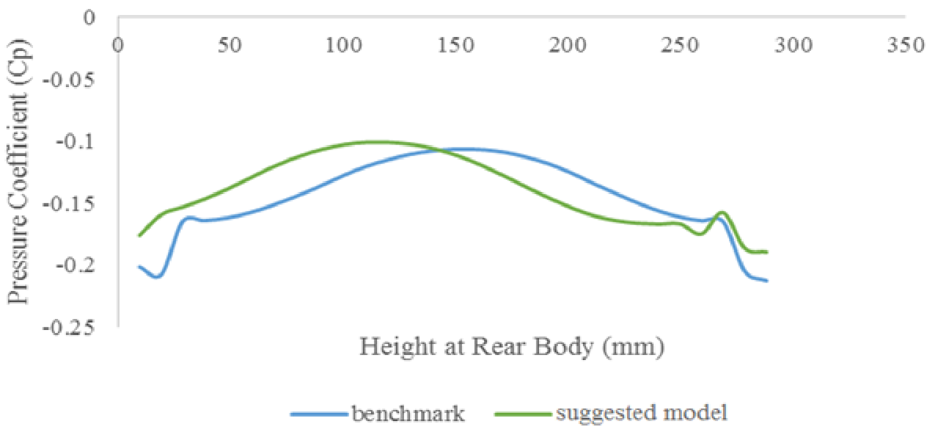


**Fig. 4.** Pressure Coefficient at Rear Bodies

## 4. Conclusions

The study shows there is a possibility to improve the aerodynamic for a bluff body vehicle. Utilizing ventury effect device create significate reduction of pressure different, reduce pressure drag and reduce energy consumption. Therefore, a bluff body shape vehicle can maintain as it is which give more space to occupant and at the same time, it is fuel efficient by having this flow correction device. The further improvement of this study is to optimize the devices so that it not only function but also decorative.

**Acknowledgement**

This work publication was financially supported by Universiti Teknikal Malaysia Melaka under grant PJP/2016/PROTON/FTK-CARE/S01508.

## References

[1] Baltas, George, and Charalabos Saridakis. "Brand-name effects, segment differences, and product characteristics: an integrated model of the car market." *Journal of Product & Brand Management* 18, no. 2 (2009): 143-151.

[2] Gilliéron, Patrick, and Azeddine Kourta. "Aerodynamic drag control by pulsed jets on simplified car geometry." *Experiments in fluids* 54, no. 2 (2013): 1457.

[3] Steininger, Karl, Caroline Vogl, and Ralph Zettl. "Car-sharing organizations: The size of the market segment and revealed change in mobility behavior." *Transport Policy* 3, no. 4 (1996): 177-185.

[4] Tamás Lajos. "Basics of vehicle aerodynamics." *Budapest University of Technology and Economics Department of Fluid Mechanics University of Rome La Sapienza* (2002).

[5] Sudin, Mohd Nizam, Mohd Azman Abdullah, Shamsul Anur Shamsuddin, Faiz Redza Ramli, and Musthafah Mohd Tahir. "Review of research on vehicles aerodynamic drag reduction methods." *International Journal of Mechanical and Mechatronics Engineering* 14, no. 02 (2014): 37-47.

[6] Lenngren, Malin, and Christoffer Håkansson. "CFD Analysis of Aerodynamic Trailer Devices for Drag Reduction of Heavy Duty Trucks." *Master's Thesis, Chalmers University Of Technology* (2010): 1-77.

[7] Huminic, Angel, Gabriela Huminic, and Adrian Soica. "Study of aerodynamics for a simplified car model with the underbody shaped as a Venturi nozzle." *International Journal of Vehicle Design* 58, no. 1 (2012): 15-32.

[8] Hassan, SM Rakibul, Toukir Islam, Mohammad Ali, and Md Quamrul Islam. "Numerical study on aerodynamic drag reduction of racing cars." *Procedia Engineering* 90 (2014): 308-313.

[9] Ahmed, Syed R., G. Ramm, and G. Faltin. *Some salient features of the time-averaged ground vehicle wake.* No. 840300. SAE Technical Paper, 1984.

[10] Franck, Gerardo, Norberto Nigro, M. Storti, and Jorge D'elia. "Numerical simulation of the flow around the Ahmed vehicle model." *Latin American applied research* 39, no. 4 (2009): 295.

[11] Johnson, Forrester T., Edward N. Tinoco, and N. Jong Yu. "Thirty years of development and application of CFD at Boeing Commercial Airplanes, Seattle." *Computers & Fluids* 34, no. 10 (2005): 1115-1151.

[12] Gmbh, F. D "Best practice guidelines for handling Automotive External Aerodynamics with FLUENT." (2005): 1–14.

[13] Chen, Shiyi, Daniel Martinez, and Renwei Mei. "On boundary conditions in lattice Boltzmann methods." *Physics of fluids* 8, no. 9 (1996): 2527-2536.

[14] Anderson, J. D. "Fundamentals of Aerodynamics." *PLoS Computational Biology* 1984, (1991).