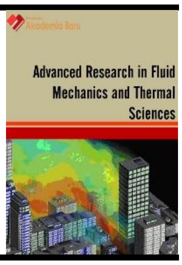
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

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



Optimization of Boundary Layer Separation Reduction Induced by The Addition of a Dimple Grid on Top of a Bluff Body

Open Access

Mohd Faruq Abdul Latif[1,*], Muhammad Nur Othman[1], Qamar Fairuz Zahmani[1], Najiyah Safwa Khashi'ie[1], Beh Eik Zhen[1], Mohd Farid Ismail[1], Ahmad Yusuf Ismail[2]

[1] Faculty of Engineering Technology, Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia
[2] Department of Mechanical Engineering, Adhi Tama Surabaya Institute of Technology, Jalan Arief Rachman Hakim, Klampis Ngasem, Sukolilo, Kota SBY, Jawa Timur 60117, Indonesia

ARTICLE INFO

Article history:
Received 16 April 2019
Received in revised form 20 September 2019
Accepted 23 October 2019
Available online 28 December 2019

ABSTRACT

In recent years, there has been an increasing interest bluff body vehicle aerodynamic. Bluff body vehicle is the vehicle segment category comprising buses, van, multi-purpose vehicle (MPV), small utility vehicle (SUV), truck and lorry. The bulky size of these bluff body experienced a large aerodynamics drag when travelling at high speed. These aerodynamic drags are contributed by large pressure drag at front and boundary layer separation on vehicle wall and wake region at the back of the vehicle. It is becoming increasingly difficult to ignore the effect of boundary layer separation on top of the bluff body towards the aerodynamic drag. Recent developments in the field of vehicle aerodynamic have led to a renewed interest in boundary layer separation phenomena on bluff body. Most studies in boundary layer separation have only been carried out on the wake region at the rear of the bluff body due to shape limitation of the bluff body itself. This research will focus on examine the boundary layer separation on top of bluff body by the adaptation of golf ball dimple grid which will be introduced on top of a generic bluff body in various location. The dimple grid will be further optimize using design of experiment (DOE) approaches. The computational fluid dynamic (CFD) simulation method is one of the more practical ways of validating the bluff body model and this will be further validate using physical wind tunnel experimental. It is possible to hypothesize that the dimple depth, size, grid pattern and location most likely have influential effect to the boundary layer separation. In general, therefore, it seems that drag reduction was possible by the alteration of flow on top of the bluff body. The optimized design shows prominent result as it contributes around 36.5% of drag reduction.

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

* *Corresponding author.*
E-mail address: faruq@utem.edu.my (Mohd Faruq Abdul Latif)

1. Introduction

The large size of the bus experienced a large aerodynamics drag when travelling in long distance at high speed. Normally, drag force can be divided into 2 types which are pressure drag and friction drag. Fiction drag is related to the boundary layer is induced which the front and rear parts has a huge pressure difference and this will affect the fuel consumption [1-2]. According to research obtained, when a vehicle is travelling in 100 km/h, the aerodynamics drag caused the heavy vehicle consume about 65% of the total energy consumption and for medium-size sedan it caused it to consumes around 20% of the fuel consumption [3-4].

There are two main factors that affect drag coefficient of bluff body which are the roundness of its front corner and the flow separation at its rear end. Hence, in this research we just focus on the flow separation which are reducing the boundary layer or the wake. Based on the stimulation results showed that by adding a designed dimple on vehicle roof and engine cover able to reduce the drag coefficient up to 10.31% when the vehicle is travelling in the speed of 30 m/s [5]. This is due to the dimples able to create an early transition to turbulent boundary layer and this helps the flow attached to the surface and reducing the boundary [6]

The increasing of fuel cost and environment burden on fossil fuel should widely concerned in automotive field. Based on this research, reducing the drag coefficient able to help the automotive manufacture company to design a vehicle with least fuel consumption by adding a passive drag device without changing the exterior design which are complicated and costly. The vision of this research is to help local bus company to reduce the fuel consumption of each buses by adding the drag reduction devices.

2. Methodology

2.1 Benchmark Ahmed Body Model Development

A basic vehicle model called Ahmed Body is developed as shown as Figure 1 as a benchmark model in this project. Ahmed Body is the simplest car model created by Ahmed *et al.,* [7] in 1980's to investigate the behaviour of newly developed turbulence models for the complex geometry cases.

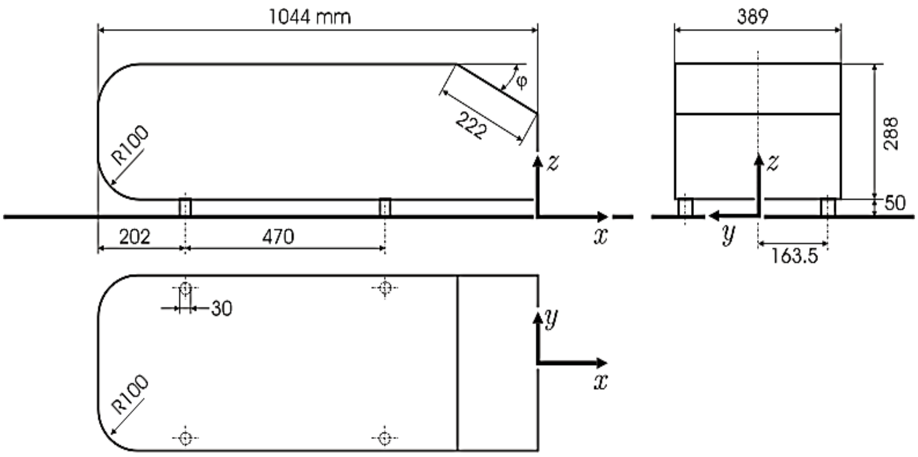


Fig. 1. The standard geometry of Ahmed Body

However, in this research some changes have been done to parameter of the Ahmed Body. The Ahmed Body model with no back slant angle was created using CATIA V5R21 software as shown as Figure 2.

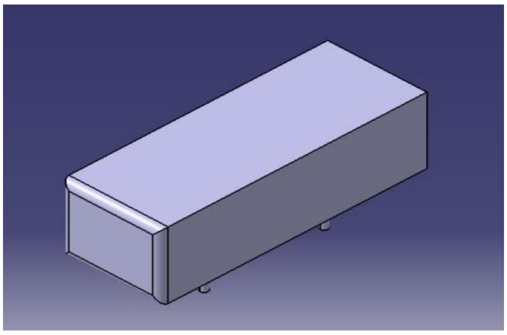


Fig. 2. The geometry development of benchmark model of Ahmed Body

2.2 Wind Tunnel Geometry Design

The wind tunnel wind tunnel geometry design should be at least two to three vehicle lengths of space in front of the car and at least five vehicle lengths behind the vehicle [8]. Based on *Wang et al.,* [4], the wind tunnel has the gap between the wind tunnel and the model at least two to three times the dimension of the model in that respective dimension. In this research, a half body simulation is planned to carry out in this research. Thus, the wind tunnel dimension will be cut half with the dimension of length L = 8.352 m, Width W = 0.935 m, and the height H = 1.400 m. The wind tunnel geometry is shown as Figure 3.

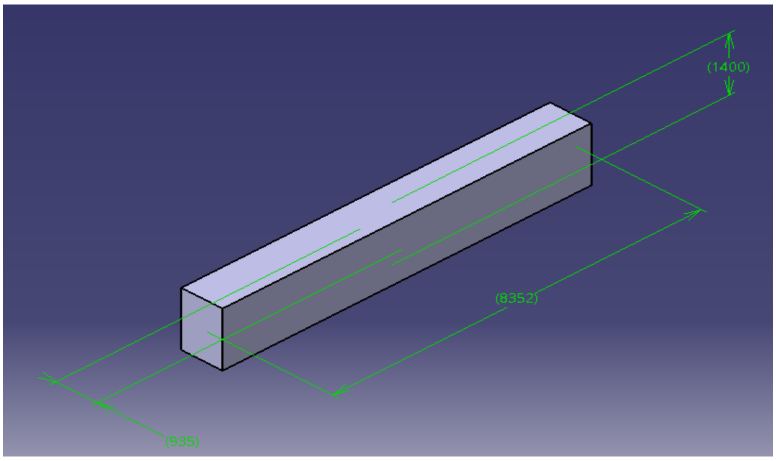


Fig. 3. The dimensions of the wind tunnel

Since half body simulation is planned to carry out in this research. Therefore, Boolean's operation is carried out to combine and removed half of the Ahmed Body part from the wind tunnel part as shown as Figure 4.

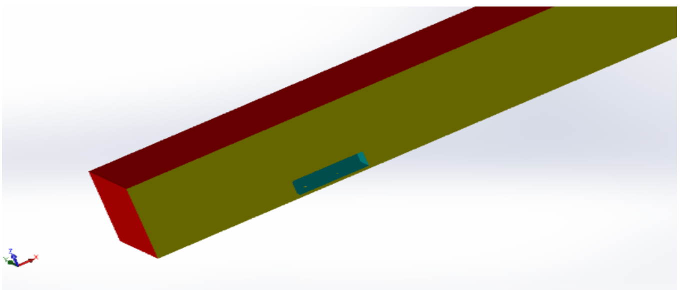


Fig. 4. Boolean operation effect between the wind tunnel and Ahmed Body part

2.3 AcuSolve Virtual Wind Tunnel Setup

Before carry out the wind tunnel test, each surface of the wind tunnel model and Ahmed Body were indicated as inflow surface, outflow surface, slip, ground, body, and symmetry. The Spalart Allmaras's turbulence equation was applied to indicate the turbulence flow. The, the inflow free stream velocity of 30.56 m/s with the eddy viscosity of 1e-5 was defined as the inlet-velocity which is same as 110 km/h speed allowed in Malaysia [9]. Surface meshing of Ahmed body are generated with 5 mm of triangular elements (1 mm absolute size elements is mesh in dimple surface geometry when the modified model is generated).

2.4 Design of Experiment (DoE)

DoE able to identify important interaction between the input factors and its effect at the same time and all possible combination can be investigated. A two-level factorial of experiment and four type of factors are developed in this experiment. Each factor has 2 types of parameters and in this research just focus on the dimension of the dimples position shifting. Other of the factors and its parameter will just have obtained from the journals related. The factor variables are tabulated in Table 1. Since 4 variables of 2 level factorial design was used. Therefore, there are total 16 different type of combination of factor is generated as shown as Table 2 and as illustrated in Figure 5.

Table 1
The input level of each variables

Factors / Variables	Low Level (-)	High Level (+)
A	104.4 mm	313.2 mm
B	29.7 mm	39.7 mm
C	4.06 m^2	12.18 m^2
D	2 mm	4 mm

**Note:* A = Dimension of Dimple Shifted, B = Diameter of Dimple, C = Dimple Coverage Area, D = Depth of Dimple

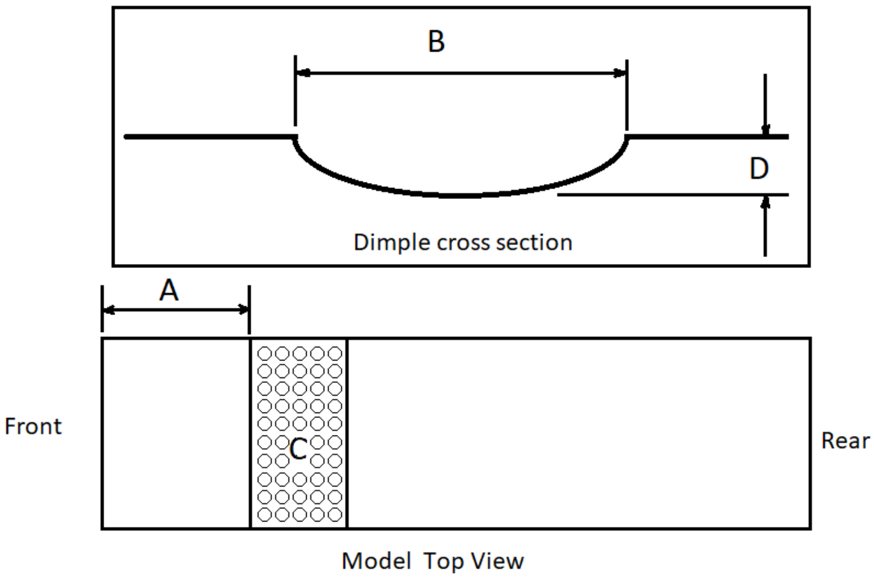


Fig. 5. Dimple diagram

Table 2
Matrix design of experiment

No	A	B	C	D
1	-	-	-	-
2	+	-	-	-
3	-	+	-	-
4	+	+	-	-
5	-	-	+	-
6	+	-	+	-
7	-	+	+	-
8	+	+	+	-
9	-	-	-	+
10	+	-	-	+
11	-	+	-	+
12	+	+	-	+
13	-	-	+	+
14	+	-	+	+
15	-	+	+	+
16	+	+	+	+

3. Results

3.1 Design of Experiment (DoE)

Based on the Figure 6 and Figure 7, it showed that all the points lie around the normal line and according to the Pareto chart which show in Figure 8; it can clearly see that the factor of dimension of dimple shifted, dimple coverage area and depth of dimple is over the t-Value limit which means those factors are significant to the response of drag coefficient.

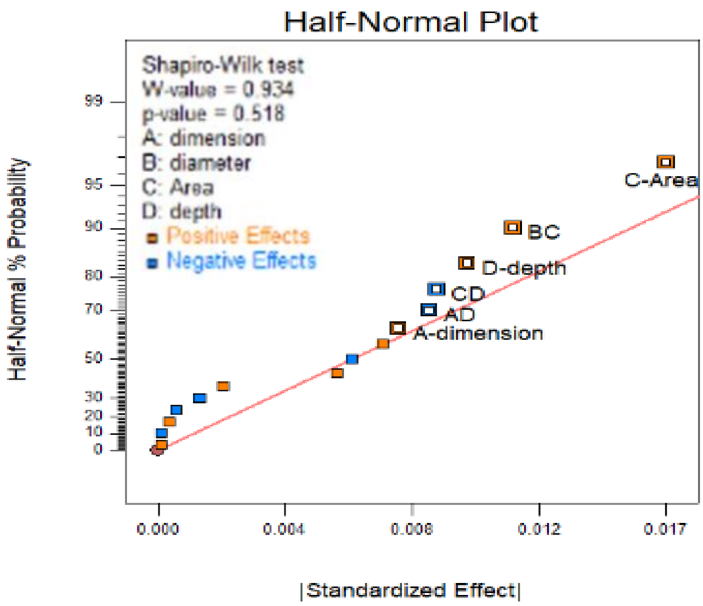


Fig. 6. The half-normal plot

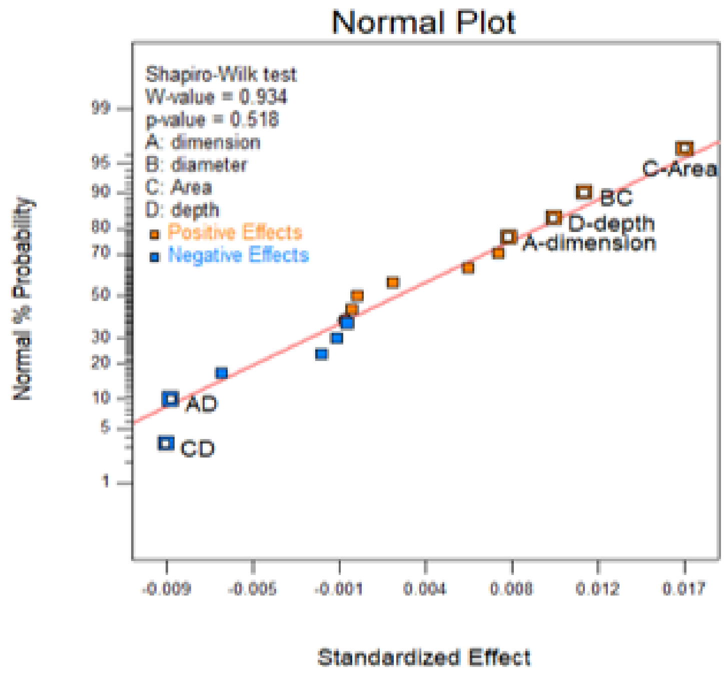


Fig. 7. The normal plot

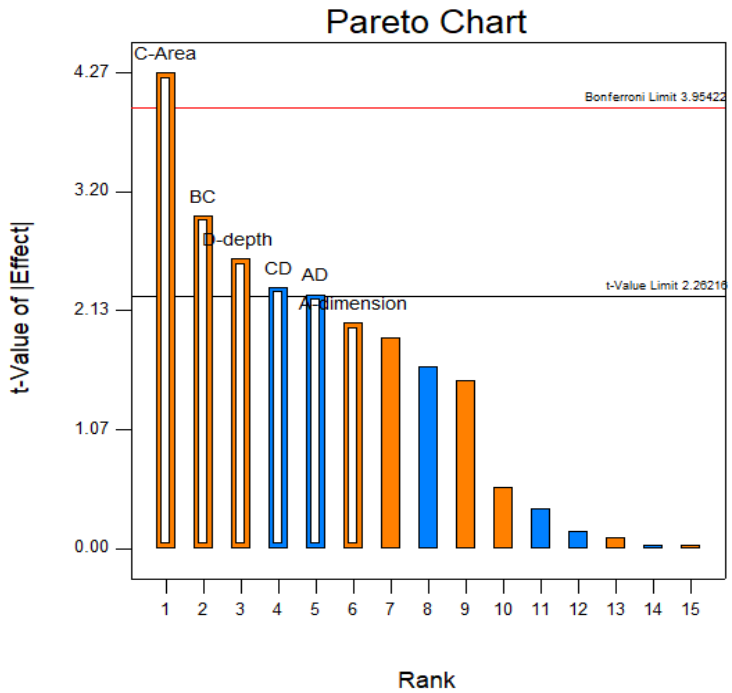


Fig. 8. The Pareto chart

3.2 Optimization Results

The numerical optimization is carried out to get the best model and there are some optimization criteria can be set as shown as Table 3. The top 5 value of desirability will be chosen and from that top 5 solution; the one solution which has the desirability value that closest to 1 will be the optimization result as shown as the colour region in Table 4.

Table 3
Optimization Criteria Setting

Factor	Goal	Lower Limit	Upper Limit	Importance
A	is in range	104.4	313.2	3
B	is in range	29.7	39.7	3
C	is in range	4.06	12.18	3
D	is in range	2	4	3
C_d	minimize	0.205	0.262	4

Table 4
The input level of each variables

No.	A	B	C	D	C_d	Desirability
1	104.400	39.700	4.06	2.000	0.211	0.894
2	105.102	39.659	4.06	2.000	0.211	0.892
3	104.405	39.591	4.06	2.001	0.211	0.891
4	105.829	39.700	4.06	2.005	0.211	0.890
5	104.400	39.476	4.06	2.000	0.211	0.889

3.3 Optimized Model of Ahmed Body

The Ahmed Body has the high pressure acting on it and the air become stagnant when it collides on the frontal area of the model which caused the high pressure acting on it. The higher-pressure air then gets separated and flow to upper and lower surface of Ahmed Body. The air after flow through the frontal area then are accelerated and this caused the pressure to drop as shown in Figure 9.

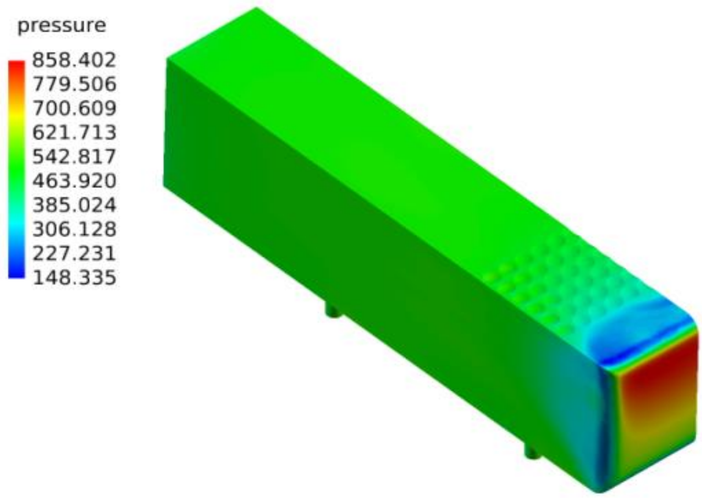


Fig. 9. The pressure contour of optimized model for Ahmed Body

Based on the Figure 10, the lower velocity on the top surface of the Ahmed Body due to the boundary layer separation at the edge of the nose of the frontal surface of the Ahmed body created a small wake region and then the air flow detach back to the Ahmed Body's top surface. When the air continue flow to the rear part of the model, the notch caused the vacuum space to occur and it creates a large wake region behind the Ahmed Body's rear part.

By comparing to the benchmark model show in Figure 10 and Figure 11, the wake region formed at front top surface is reduced when the dimple is added. This is due to the tinny pocket turbulence is induced. The airflow pressure inside the dimple area increase and when then airflow continue flow out from the dimple, the pressure slightly decreases and this caused the airflow always attach to the surface of the dimple [10].

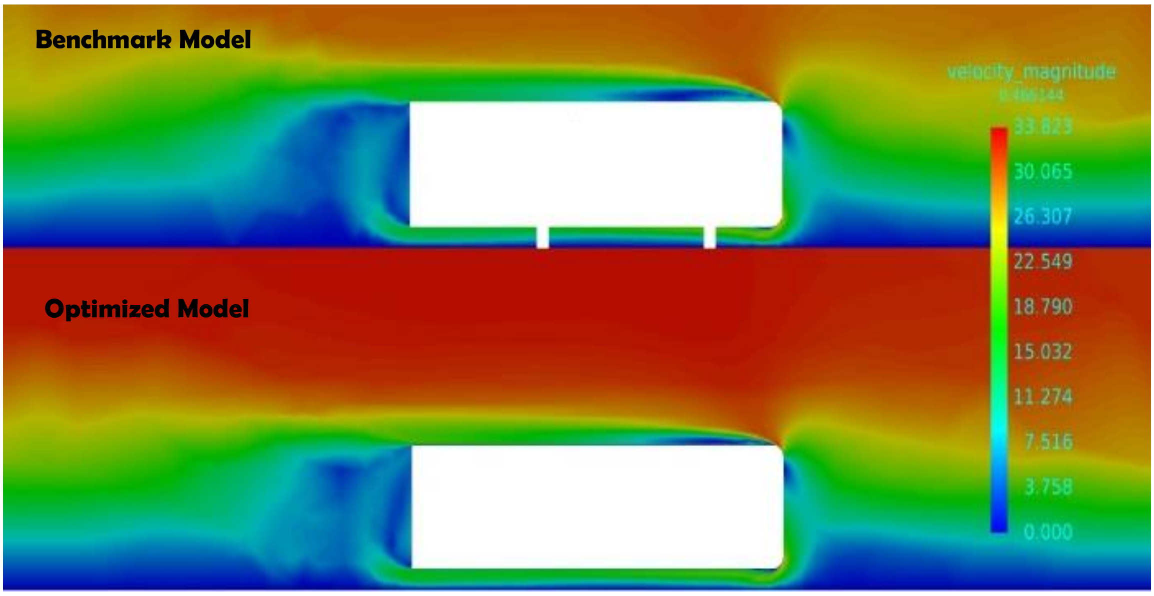


Fig. 10. The velocity magnitude contour of benchmark and optimized model Ahmed Body

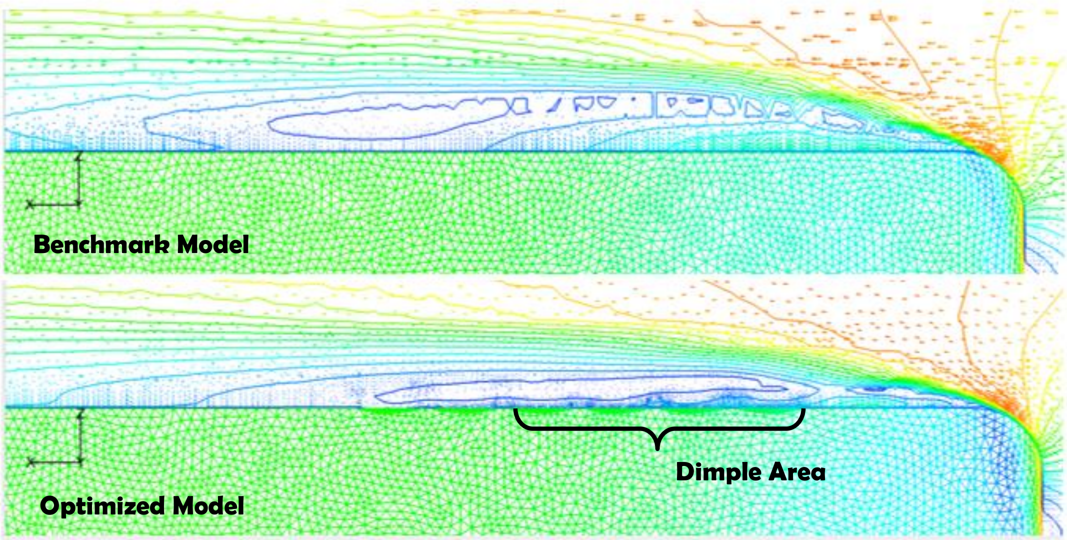


Fig. 11. The velocity magnitude contour of benchmark and optimized model Ahmed Body

Figure 12 showed the comparison between the pressure coefficients along the surface of the Ahmed Body without adding any dimples and with dimples. At the top surface of the upper body, lower in pressure gradient value resulting to the reduction of boundary layer separation. Hence, drag coefficient value will be lower. The greater the pressure gradient, the greater the boundary layer separation.

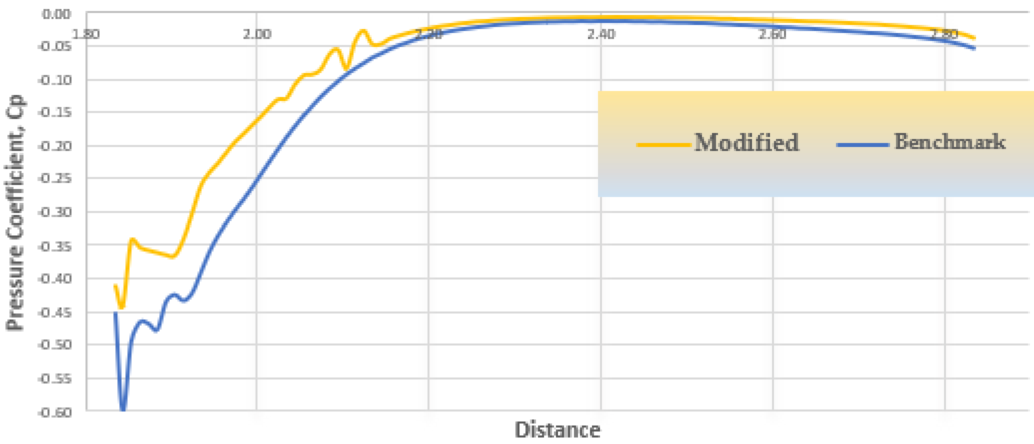


Fig. 12. The graph of the comparison of pressure coefficient

3.4 Fuel Consumption

Assuming that the velocity of the coach bus travelled, v: 30.56 m/s (110 km/h), The frontal area of the coach bus, A: 9.69 m^2. Mass of the real coach bus, m: 16000 kg. Percentage of Fuel Reduction due to reduction of drag coefficient is 18.58%.

Since the drag reduction is about 36.5%, if the optimized model's design is applied to the real buff body vehicle such as coach bus; the fuel reduction for the coach bus will be increase up to 18.54% (Table 5).

Table 5
Comparison of Fuel Consumption based on Optimized Drag Coefficient

Model	Benchmark	Optimized
Drag Coefficient, C_D	0.52	0.33
Drag Force, D	2.88 kN	1.83 kN
$D = 0.5\,\rho v^2 A C_D$		
Power Required to overcome the drag force, P_D	88.08 kW	55.90 kW
$P_D = 0.5\,\rho v^3 A C_D$		
Rolling Resistance, F_R	1.57 kN	1.57 kN
$F_R = m \times g \times C_r$		
Power Required to overcome the rolling resistance, P_R	48 kW	48 kW
$P_R = m \times g \times C_r \times v$		
Transmission Power losses, P_T	9.6 kW	9.6 kW
$P_T = 0.2 \times P_R$		
Fuel used to overcome aerodynamic drag, G	0.302	0.246
$G = 0.5\left(\frac{P_D}{P_D + P_R + P_T}\right)$		

4. Conclusions

As a conclusion, the implementation of dimple on top surface of Ahmed body has reduce 36.5% drag coefficient value at position 104.4 mm from front top surface of Ahmed Body. When the dimple is at corresponding position on top surface of upper body. This gives an outcome fuel savings at around 18.54%.

5. Recommendation for Future Work

Since Ahmed body is the simplest wind tunnel model used to represent the ground vehicles, in order to improve the quality of the simulation result; a real model of vehicle design is recommended to use in virtual wind tunnel test in future work. Since the real vehicle model geometry has the similar geometry and layout design to the real vehicle, hence it able to track and investigate the airflow along the vehicle more reality and its results are more accurate. Besides that, underbody flow is one of the factor that effect the aerodynamics drag. There is about 30% of the total drag which is origin from the underbody flow [11]. A little modification can be carried out to control the underbody flow such as undercarriage straight skirt, it able to block the lateral flow from the side wind through the gap between ground and the vehicle body [1].

Acknowledgement
The author would like to express appreciation to Vibro Acoustics Research Group, Centre of Research on Energy (CARe) and Universiti Teknikal Malaysia Melaka (UTeM) for funding this project under PJP/2016/PROTON/FTK-CARE/S01508

References
[1] Martín-Alcántara, A., E. Sanmiguel-Rojas, C. Gutiérrez-Montes, and C. Martínez-Bazán. "Drag reduction induced by the addition of a multi-cavity at the base of a bluff body." *Journal of Fluids and Structures* 48 (2014): 347-361.
[2] Chear, C. K., and S. S. Dol. "Vehicle aerodynamics: drag reduction by surface dimples." *International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering* 9, no. 1 (2015).
[3] Jing, Chen, Wu Yun-zhu, Wang Deng-feng, Xu Wei-gang, and Wen Gui-jin. "Research on aerodynamic drag reduction of heavy commercial vehicle." In *2010 2nd International Conference on Computer Engineering and Technology*. 2010.
[4] Wang, Yiping, Cheng Wu, Gangfeng Tan, and Yadong Deng. "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* 231, no. 1 (2017): 130-144.
[5] Song, Xiao-wen, Guo-geng Zhang, Yun Wang, and Shu-gen Hu. "Use of bionic inspired surfaces for aerodynamic drag reduction on motor vehicle body panels." *Journal of Zhejiang University-SCIENCE A* 12, no. 7 (2011): 543-551.
[6] Toet, Willem. "Aerodynamics and aerodynamic research in Formula 1." *The Aeronautical Journal* 117, no. 1187 (2013): 1-26.
[7] Khan, Rehan Salahuddin, and Sudhakar Umale. "CFD aerodynamic analysis of Ahmed body." *International Journal of Engineering Trends and Technology* 18, no. 7 (2014): 301-308.
[8] Lanfrit, Marco. "Best practice guidelines for handling Automotive External Aerodynamics with FLUENT." (2005).
[9] Latif, MF Abdul, M. N. M. Zain, and M. Azwar Azhari. "The introductory of dimple to the Ahmed body top surface as a drag reduction device." *Proceedings of Mechanical Engineering Research Day 2017* 2017 (2017): 64-65.
[10] Lahaye, A., Leroy, A., Kourta, A. “Characterisation of a square back Ahmed body near wake flow.” *21eme Congres Fran¸ cais de Mecanique Bordeaux* (2013): 1-6.
[11] Wood, Richard M. "A discussion of a heavy truck advanced aerodynamic trailer system." In *Int. Symp. Heavy Veh. Weights Dimens., 9th, University Park, PA*. 2006.