

EXPERIMENTAL STUDY ON THE INFLUENCE OF RIBLETS ON BUS BODY TOWARDS DRAG COEFFICIENT

Matthew Sebastian A.¹⁾, Fandi D. Suprianto²⁾, Sutrisno Teng³⁾

Mechanical Engineering Department Petra Christian University^{1,2,3)}

Jl. Siwalankerto 121-131, Surabaya 60236. Indonesia^{1,2,3)}

E-mail : matthew.sebastian.77@gmail.com¹⁾, fandi@petra.ac.id²⁾, tengsutrisno@petra.ac.id³⁾

ABSTRACT

Heavy vehicles such as buses and trucks, generally have a form that is not very aerodynamic, usually very long and boxy. Aerodynamics can be very influential on the use of fuel when the coefficient of drag is reduced, the vehicle can be more efficient in their use of fuel because the force that inhibits will be reduced. One way to reduce this drag force is with riblets. Riblets is a structure shaped like a bunch of lined trenches. The results of the study on flat plate showed that with the riblets, up to 10% drag reduction can be achieved with the blade structure compared with only a flat surface under the same conditions.

Methods to validate the experiments were performed using simulation results with experimental results from the wind tunnel at a speed of 16-27 m/s. Bus model created by CAD software according to its original shape to be tested in the wind tunnel and then compare it with the results of the ANSYS Fluent simulation. Riblets geometry studied using ANSYS Fluent simulation and the results of the Boundary Layer will be observed as the riblets geometry changes. From the research, it is known that the best riblets geometry is 1% of the overall length of the bus and with the application of said geometry, 1.46% drag reduction can be obtained, from Cd 0.548 to 0.54. The model with the best drag reduction uses riblets on the front roof of the bus.

Key Words:

Coefficient of drag, Riblets, Wind Tunnel, Boundary Layer, Aerodynamics.

1. Preface

Heavy vehicles such as buses and trucks, generally have a form that is not very aerodynamic, tend to be long and large in size. Drag forces or aerodynamic force is a force that hinder the movement of a vehicle through a fluid (liquid or gas). Therefore, aerodynamics can greatly affect fuel consumption for vehicles because, vehicles with smaller drag forces, the vehicle can be more efficient at using fuel because the force that inhibits become reduced. Besides aerodynamics may also affect the stability of long vehicles such as buses and trucks.

Drag forces, (friction drag in particular) is drag caused by friction of the fluid against the surface of an object in parallel. The longer the vehicle, friction drag will increase. This is because the amount of surface that is parallel to the wind direction compared with ordinary cars like sedan or hatchbacks.

On the other hand, nature has its own way to reduce friction in the fluid, one example is a shark. Shark skin have a unique texture compared to other similar fish. Texture of shark skin will feel different when palpated in two different directions. This is due to the texture scales called riblets. Riblets is a structure shaped like a bunch of lined trench that extends and in the direction of fluid flow. With the texture of this riblets, the shark can reduce the friction drag and can swim faster than other fishes. This happens because riblets actually make the transision from laminar boundary layer to turbulent even faster by creating vortex inside the riblet valley.

This phenomenon previously been investigated in

2010 by Brian Dean and Bharat Bhushan at Ohio State University, USA. They begin to analyze various forms of riblets in sharks and get three kinds of structures : sawtooth, scalloped and blade. The results of this study indicate that by replicating the structure of riblets, up to 10% drag reduction can be obtained compared with the flat surface under the same conditions. This study shows that the principle of riblets is possible to be implemented in various applications. One example is a textured swimsuit with nano sized riblets proved to be superior compared to those without. Other applications is for airfoil, but there is still no application in vehicles such as cars and buses. Therefore, research is needed to understand the optimum structure and riblet placement on vehicles, especially on long vehicles such as buses.

1.1 Research Objectives

- Optimizing the shape, size, and riblets placement on the body of bus against coefficient of drag.
- Comparing the coefficient of drag between the standard bus and the modified one with riblets.

1.2 Scope of Research

- This research is only focused at Coefficient of Drag.
- Model used for research is Mercedes Benz Travego

1.3 Research Benefits

- To observe the effect of riblets application on the body of the bus against coefficient of drag
- Obtain the optimum design and placement of riblets to reduce coefficient of drag.

2. Research Methodology

The research methods that will be used to carry out this study is based on the following flowchart:

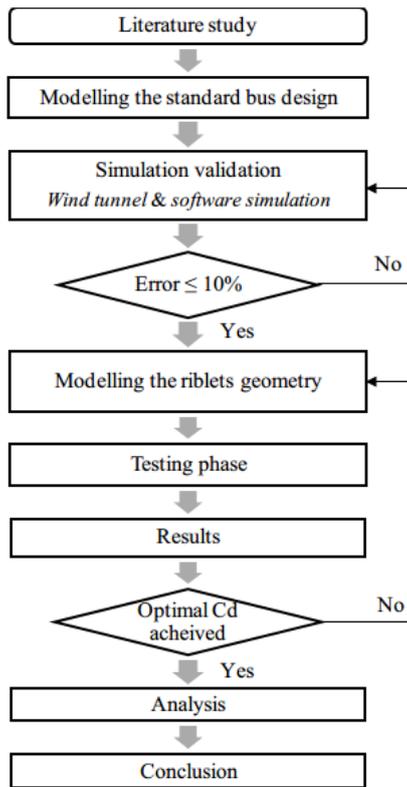


Figure 1. Research flowchart

2.1 Modelling the standard bus design

A miniature bus model needs to be created first in Solidworks. Design created from the Mercedes Travego blueprint with its original dimensions. Bus model created with 1:1 dimensions in Solidworks and then scaled down to 1:50 for 3D print. 3d model does not include tyres and side mirrors

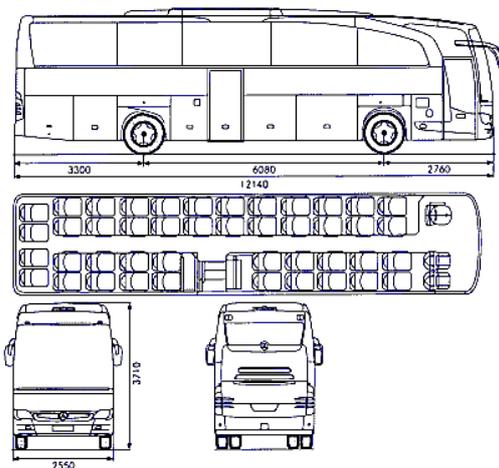


Figure 2. Mercedes Travego blueprint

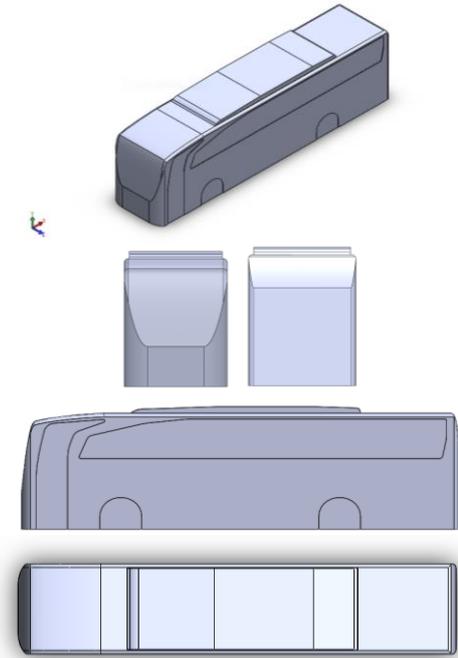


Figure 3. Mercedes Travego 3D model

Table 1. Dimension difference

Mercedes Travego Dimensions	Full scale	3D Print
Length	12140 mm	242,8 mm
Width	2550 mm	51 mm
Height	3710 mm	67,66 mm

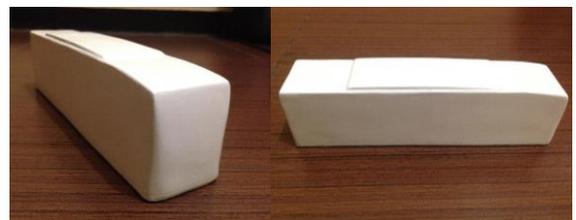


Figure 4. 3D print results

2.2 Wind Tunnel testing

To validate the wind tunnel speed, validation is performed with manual manometer and digital anemometer. Parameters used for this validation are : kerosene density = 820 kg/m^3 ; air density = $1,1649 \text{ kg/m}^3$; earth gravitational constant = $9,8 \text{ m/s}^2$; manometer angle : $\sin 15 = 0,258819045$.



Figure 5. Wind Tunnel



Figure 6. Pitot tube, manual manometer and digital anemometer

- **Pressure Calculation**

$$p = \rho \cdot g \cdot h$$

$$p = \rho \cdot g \cdot (2 \cdot \Delta l \cdot \sin \theta)$$

$$p = 820 \cdot 9,8 \cdot (2 \cdot 0,035 \cdot \sin 15)$$

$$p = 145,58 \text{ Pa}$$

- **Velocity Calculation**

$$v = \sqrt{\frac{2 \cdot p}{\rho_{\text{air}}}}$$

$$v = \sqrt{\frac{2 \cdot 145,58}{1,1649}}$$

$$v = 15,81 \text{ m/s}^2$$

Table 2. Wind tunnel speed validation

RPM	Δl mano meter (m)	P (Pa)	V (m/s)	V anemo meter (m/s)	Error (%)
800	0.03	144.19	15.73	16.03	1.92
900	0.04	185.79	17.86	18.13	1.54
1000	0.05	209.36	18.96	19.33	1.97
1100	0.06	248.18	20.64	21.10	2.23
1200	0.07	281.46	21.98	22.57	2.66
1300	0.08	334.14	23.95	24.37	1.74
1400	0.09	353.55	24.64	25.37	2.97
1500	0.10	397.92	26.14	26.83	2.66
				Average	2.21

The validation shows 2:21% difference between manual manometer and digital anemometer. This error is acceptable and this stated that the digital anemometer can be used as a valid airspeed measurement tool.

After the wind speed is validated, next step is to validate the drag coefficient results between the wind tunnel experiment and ANSYS Fluent simulation. Here are the procedures :

1. Place the model inside the wind tunnel test section and then install the model on the wind tunnel balance.



Figure 7. Bus model inside the test section

2. Ensure that the test section is airtight
3. Set the wind tunnel balance measuring needle on the scales

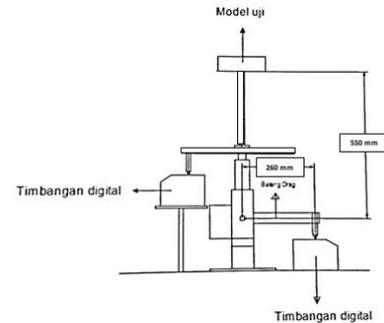


Figure 8. Wind Tunnel Balance

4. Reset the scales to zero (0).
5. Turn on the wind tunnel and set to 800 rpm
6. Obtain drag test results from the scales (mass).
7. Repeating steps no. 4-5 at 900 to 1500 rpm with an interval of 100 rpm.

Parameters used are as follows: earth gravitational constant (g) = 9,8 m/s²; air density (ρ) = 1,1649 kg/m³; bus frontal area (A) = 0,00343055 m²; moment arm drag 1 (l_1) = 0,55 m; moment arm drag 2 (l_2) = 0,26 m.

2.3 ANSYS Simulation

Bus models of the same size as 3D Print will be simulated with ANSYS Fluent. Here are the settings and parameters that will be used to simulate the bus.

2.3.1 Bus Simulation

- **Design Modeler**

- ✓ Enclosure size (X,Y,Z) : 150 mm* x 150 mm* x 475 mm *(XY plane symmetry)
- ✓ Body of influence (Y,Z) : 380.8mm x 93.97mm, Distance from inlet : 114.84mm, Extrude 60 mm

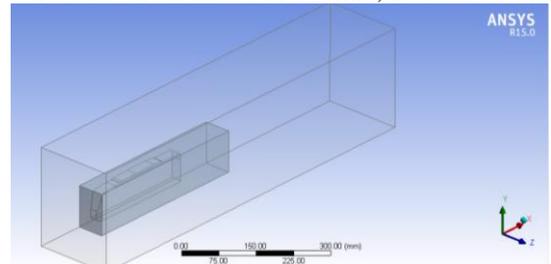


Figure 9. Bus enclosure model

- **Mesh**

Table 3. Bus Mesh settings

Sizing	
Use Advanced Size Function	On: Proximity and Curvature
Smoothing	High
Transition	Slow
Num Cells Across Gap	3
Min Size	2.e-004 m
Proximity Min Size	2.e-004 m
Max Face Size	Default (4.5529e-002 m)
Max Size	Default (9.1057e-002 m)
Growth Rate	Default (1.20)

Inflation	
Use Automatic Inflation	Program Controlled
Inflation Option	First Aspect Ratio
First Aspect Ratio	8
Maximum Layers	3
Growth Rate	1.2

After setting the mesh with table above, we are going to set the body of influence with 5 e-05 m element size. The result can be seen below :

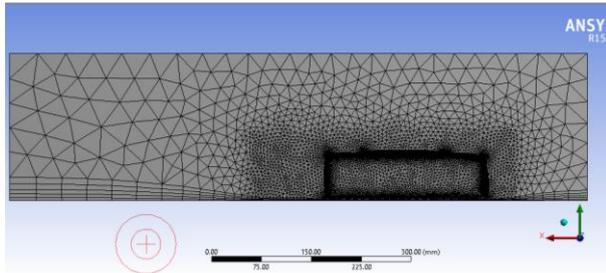


Figure 9. Mesh results

• Fluent Settings

Fluent simulation is done in 3D, steady state and uses realizable k-epsilon turbulence model. For external aerodynamics application, non-equilibrium wall treatment is chosen. For the standard bus model, the inlet velocity follows with the one from the wind tunnel testing (16-17 m/s) with 1% turbulence intensity. Simulation is done until the convergence criterion reaches below 1e-03.

Table 7. Bus Boundary Condition Settings

Wall Zones	- Velocity inlet : velocity-inlet	
	- Pressure outlet : pressure-outlet	
	- Bus body : wall	
	- Symmetry : symmetry	
	- Symmetry-side : symmetry	
	- Symmetry-top : symmetry	
- Road : wall		
Fluid Properties	Fluid Type	Air
	Density	$\rho = 1.1684 \text{ kg/m}^3$
	Viscosity	$\nu = 0.0000186 \text{ kg/m.s}$

Table 8. Bus Solution Methods

Scheme	Coupled
Spatial Discretization	• Gradient : Least Squares Cell Based
	• Pressure : Standard
	• Momentum : First Order Upwind
	• Turbulence Kinetic Energy : First Order Upwind
	• Turbulence Dissipation Rate : First Order Upwind

2.3.2 Riblets Simulation

Based on the previous studies of riblets, the best drag reduction is achieved with blade model using the geometrical ratio of $h/s = 0.5$ and $t/s = 0.02$, where s: distance between riblets, h: riblet height, and t: riblet thickness. [3]

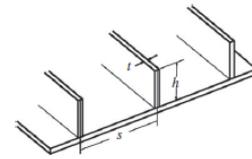


Figure 10. Riblets geometry

From this geometry, variation will be made based of the distance between riblets by 1-3% compared to the overall length of the bus. Here are the size comparison: Bus Length: 12140 mm, length of bus model: 242.8 mm.

Table 9. Riblets geometry variations

Model (mm)			
%	s	h	t
3%	7.28	3.64	0.07
2.5%	6.07	3.04	0.06
2%	4.86	2.43	0.05
1.5%	3.64	1.82	0.04
1%	2.43	1.21	0.02

Fullscale (mm)

%	s	h	t
3%	364.2	182.1	3.64
2.5%	303.5	151.75	3.04
2%	242.8	121.4	2.43
1.5%	182.1	91.05	1.82
1%	121.4	60.7	1.21

Simulation will be done with ANSYS Fluent and here are the settings used for simulation:

• Design Modeler

- ✓ Enclosure size (X,Y) : 242.8 mm x 100 mm
- ✓ Riblets test section (X) : 140 mm, Distance from inlet : 48.5 mm

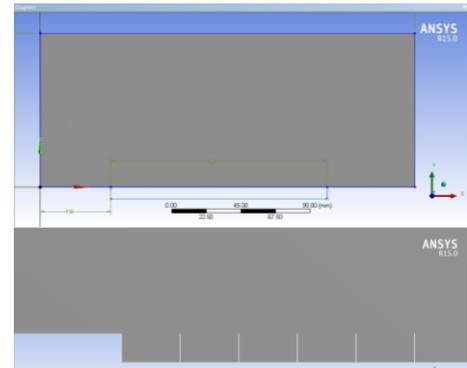


Figure 11. Riblets test section

• Mesh

Table 10. Riblets Mesh settings

Sizing	
Use Advanced Size Function	On: Proximity and Curvature
Smoothing	Medium
Transition	Slow
Min Size	Default (0.131110 mm)

Proximity Min Size	Default (13.1110 mm)
Max Size	Default (26.2230 mm)
Growth Rate	Default (1.20)

After setting the mesh with table above, we are going to set surface with mapped face meshing to get hex mesh. And then using the edge sizing function with bias enabled so the boundary layer that occurs in the surface of the plate can be captured for analysis. The result can be seen below :

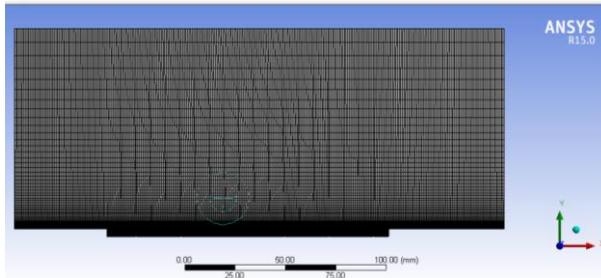


Figure 12. Riblets Mesh results

• Fluent Settings

Fluent simulation is done in 2D Double precision, steady state and uses realizable k-epsilon turbulence model. For fine mesh application, enhanced wall treatment is chosen. For the experiment, the inlet velocity is 23 m/s with 1% turbulence intensity. Simulation is done until the convergence criterion reaches below 1e-06. Spatial discretization is the same as the bus simulation.

Table 11. Riblets Boundary Condition Settings

Wall Zones	- Inlet : velocity-inlet
	- Outlet : pressure-outlet
	- Plate : wall
	- Far_field : symmetry

After the simulation is done, boundary layer is going to be observed at the same point for every variation which is 0.227 m from the inlet. The boundary layer can be obtained by iso-surface and then exporting the result on a Microsoft excel file. The shape factor of the boundary layer can be calculated with displacement thickness and momentum thickness equations.

3. Results and Analysis

In this study the addition of riblets on this bus, the first stage that is carried out after the preparation of standard bus body is the drag coefficient measurement and validate it with ANSYS simulation results. The second stage is to research the optimum geometry of riblets. The last step is the application of the best riblets geometry to the roof of bus body that will be compared to the standard model of bus.

3.1 Standard bus simulation validation

The wind tunnel data is calculated with parameters explained in the previous chapter and then compared with the simulation results that fulfill all the convergence criterion previously explained. Here are the results :

Table 12. Coefficient of Drag Comparison

v (m/s)	Cd (Wind Tunnel)	Cd (ANSYS)
16	0.58	0.58
18	0.56	0.57
19	0.58	0.57
21	0.55	0.56
23	0.56	0.55
24	0.59	0.54
25	0.59	0.54
27	0.58	0.53
Average	0.57	0.56

From the average results, it can be calculated how much difference / error of the standard bus coefficient of drag by using the following formula:

$$\text{error} = \frac{C_{D \text{ wind tunnel}} - C_{D \text{ simulation}}}{C_{D \text{ wind tunnel}}} \times 100\%$$

$$\text{error} = \frac{0.5746 - 0.5551}{0.5746} \times 100\%$$

$$\text{error} = 3.39\%$$

The average coefficient of drag difference between the ANSYS simulation results and wind tunnel is 3:39%. The difference is considered good enough (below 5%), and it can be concluded that the ANSYS simulation results can be used for subsequent experiments.

3.2 Riblets geometry simulation results

With the riblets variation on previous chapter, testing parameters is done in accordance with section 2.2. The final results that will be used as a comparison is Shape Factor (H). The shape factor desired value is the smallest among the five variations riblets that have been determined. Parameters for testing is the air speed at 23 m/s, equivalent to 82.8 km / h is taken as the average speed of buses. Below is the shape factor results :

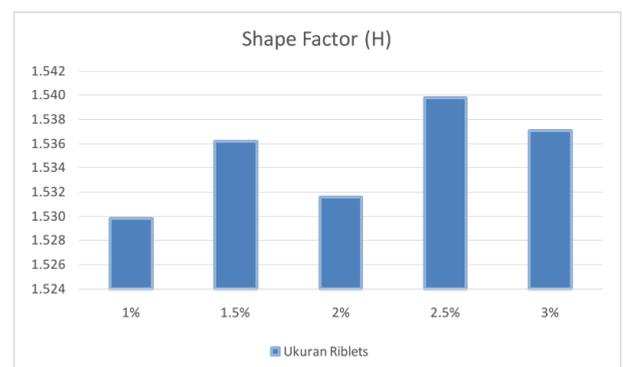


Figure 13. Shape factor results

From the chart above, it can be concluded that the 1% riblets width of the overall length of the bus has the best shape factor (1.530). This was followed by riblets 2% which is 1,532, 1.5% with 1,536, 3% with 1,537, and the last one is the size 2.5% with shape factor of 1,540. The desired shape factor is the smallest because due to a smaller value, drag reduction can be generated even greater. This is done by speeding up the boundary layer transition from laminar to turbulent by riblets as a vortex

generator. With that results, riblets is proven to accelerate the transition process of the state of the laminar boundary layer to turbulent (shape factor $H = 1.5$). From the above table can also be seen that the size of each riblets variaion have an erratic shape factor values and do not have a regular trend. This is because the characteristics of the boundary layer which is sensitive to riblets geometry, so small differences in geometry will give different shape factor results.

3.2 Riblets application on bus body and analysis

Following the riblets geometry experiment analysis, the best result is obtained with 1% width (s) of the overall length of the bus. This size will be used on the roof of the bus body. In this experiment, 3 kinds of riblets placement arrangement are made as follows:

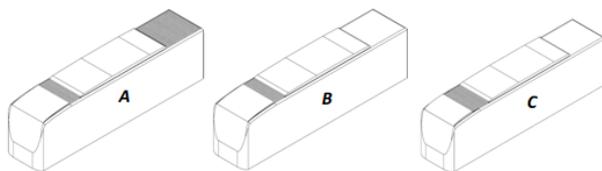


Figure 14. Bus A, Bus B and Bus C

- ✓ Bus A : riblets on the front and the back

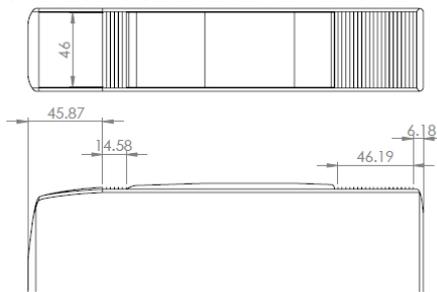


Figure 15. Bus A riblets placement

- ✓ Bus B : riblets on the front only

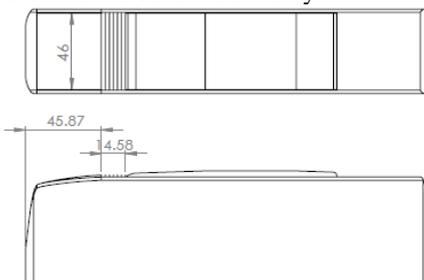


Figure 16. Bus B riblets placement

- ✓ Bus C : same as Bus B but with extra 4 riblets

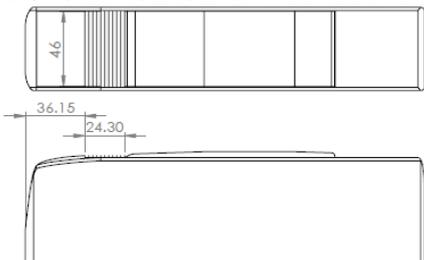


Figure 17. Bus C riblets placement

Every bus model is simulated with ANSYS Fluent at 23 m/s windspeed. Here are the example of velocity contour on the standard bus model :

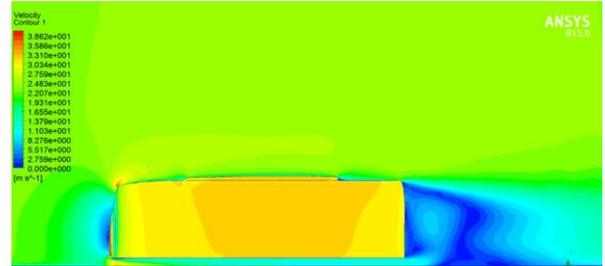


Figure 18. Standard bus velocity contour

From the picture, the velocity contour can be analyzed of what is happening on the roof of the bus and also the comparison of the magnitude of wake behind the bus. Contours of velocity that occur in the roof of the bus will be discussed below:

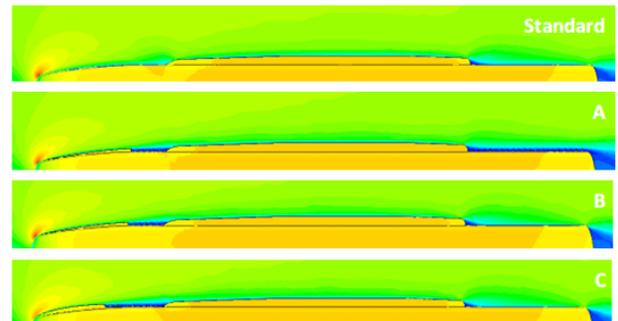


Figure 19. Velocity contour comparison

Of the four images above, it can be observed that with the presence of riblets on the roof of the bus, the contours of velocity that occurs is smoother than the standard bus. This is due to a condition in which the airflow in standard bus will hit the AC module in advance that tends to be higher and prominent. With the presence of riblets as a vortex generator, the airflow is more attached than standard conditions. Furthermore, it can be observed that there is a bubble separation at the back of the AC module where the flow path is moving away from the roof of the bus (separated). This is caused by the shape of the AC module that is higher and protrudes from the roof of the bus. On Bus A which has riblets on the back of the AC module, it appears that placement is less than ideal for drag reduction. This is because the dimensions of the bus is long and the airflow at the rear will lose momentum. With the riblets in the back, then the air flow will be impeded and caused larger wake than standard.

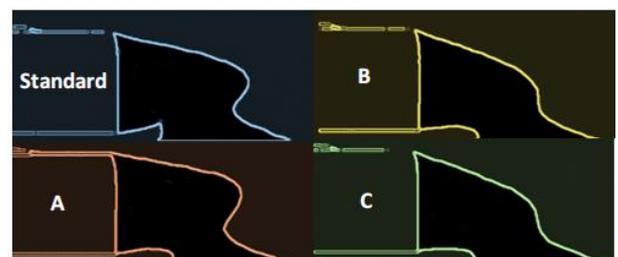


Figure 20. Wake comparison



Figure 21. Combined wake comparison

Of the four images above, it can be seen that when compared to standard buses, bus A has the biggest wake. This is due to the placement of riblets on the back that tends to inhibit the flow of air, as already discussed before. For bus B and C, wake generated tend to be smaller. This proves that with the addition of riblets, air flow will remain attached to the body of the bus and eventually has an impact on the reduction of the drag coefficient of the bus. All the results of the four simulated drag coefficient bus models above can be compared in the table below:

Table 13. Coefficient of drag comparison with standard bus model

Bus Model	Cd	% difference
Standard	0.548	-
Bus A	0.565	- 3.1 %
Bus B	0.540	1.46 %
Bus C	0.542	1.1 %

From the results of the table above it can be seen that the arrangement of riblets which have the greatest drag reduction is the model Bus B at 1:46%. This was followed by Bus C with 1.1% drag reduction. Bus Model A does not fix the coefficient of drag of the standard bus model because of riblets at the back which tends to inhibit the flow of air. This leads to a larger wake than a standard bus conditions. Selection of the arrangement and placement of riblets have a big impact on coefficient of drag and from this experiment, it was found that the fewest number of riblets is the best in this experiment (Bus B).

4. Conclusion

From the research that has been done in previous chapters such as testing riblets and buses drag coefficient, the experimental results can be summarized as follows:

- Based on the ANSYS Fluent results using riblets geometrical ratio obtained from previous stuides ($h/s = 0.5$ and $t/s = 0:02$, where s : distance between riblets, h : height riblets, and t : thickness riblets) [3], the best shape factor is generated by the geometry of riblets with a width of 1% of the total body length of the bus (1.53).
- The roof of the bus modified with 3 riblets placements model namely Bus A with riblets on the front and rear, Bus B with the front riblets only, and

Bus C that is the same as Bus B with the addition of 4 pieces of riblets.

- Experiments for 3 bus models above performed with ANSYS Fluent and then compared with the standard bus model (Cd: 0548) at a speed of 23 m/s. The bus model with the best drag reduction is Bus B at 1:46% (Cd: 0.54), followed by Bus C at 1.1% (Cd: 0.542). Bus Model A does not fix the coefficient of drag of a standard bus model because the riblets placement behind tends to inhibit the flow of air and causing a bigger wake to be generated (Cd: 0565).

References

1. ANSYS Inc. (2009). *Meshing Help*.
2. ANSYS Inc. (n.d.). *Introduction to ANSYS FLUENT : Turbulence Modeling*.
3. Bhusan, B., & Dean, B. (2010). Shark-skin surfaces for fluid drag reduction in turbulent flow : a review. *Royal Society Publishing Journal*. Retrieved November 19, 2015
4. Cengel, Y. A., & Cimbala, J. M. (2006). *Fluid Mechanics Fundamentals and Applications*. McGraw-Hill.
5. de Souza, A. (2005). *How to - Understand Computational Fluid Dynamics Jargon*. NAFEMS Ltd.
6. El Gharbi, N., Absi, R., Benzaoui, A., & Amara, E. (2009). Effect of near -wall treatments on airflow simulations. *Computational Methods for Energy Engineering and Environment*.
7. Fox, R. W., McDonald, A. T., & Pritchard, P. J. (2011). *Introduction to Fluid Mechanics*. Wiley.
8. Hucho, W.-H. (1987). *Aerodynamics of Road Vehicles*. Elsevier.
9. Lanfrit, M. (2005). *Best practice guidelines for handling Automotive External Aerodynamics with FLUENT*. Germany: Fluent Deutschland GmbH. Retrieved 05 21, 2016, from http://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Ext_Aero_Best_Practice_Ver1_2.pdf
10. Mohamed, E. A., Radhwi, M. N., & Gawad, A. F. (2014). Computational investigation of aerodynamic characteristics and drag reduction of a bus model. *American Journal of Aerospace Engineering*.
11. Mokhtar, W., & Hasan, M. (2016). A CFD study of Wind Tunnel Wall Interference. *American Society for Engineering Education*.