Wind and Structures, Vol. 11, No. 4 (2008) 257-273
DOI: http://dx.doi.org/10.12989/was.2008.11.4.257

Aerodynamics of an intercity bus

Rajnish Sharma*, Daniel Chadwick and Jonathan Haines

Department of Mechanical Engineering, The University of Auckland, Private Bag 92019, Auckland, New Zealand

(Received October 17, 2006, Accepted May 2, 2008)

Abstract. A number of passive aerodynamic drag reduction methods were applied separately and then in different combinations on an intercity bus model, through wind tunnel studies on a 1:20 scale model of a Mercedes Benz Tourismo 15 RHD intercity bus. Computational fluid dynamics (CFD) modelling was also conducted in parallel to assist with flow visualisation. The commercial CFD package CFX^{TM} was used. It has been found that dramatic reductions in coefficient of drag (C_D) of up to 70% can be achieved on the model using tapered and rounded top and side leading edges, and a truncated rear boat-tail. The curved front section allows the airflow to adhere to the bus surfaces for the full length of the vehicle, while the boat-tails reduce the size of the low pressure region at the base of the bus and more importantly, additional pressure recovery occurs and the base pressures rise, reducing drag. It is found that the CFD results show remarkable agreement with experimental results, both in the magnitude of the force coefficients as well as in their trends. An analysis shows that such a reduction in aerodynamic drag could lead to a significant 28% reduction in fuel consumption for a typical bus on intercity or interstate operation. This could translate to a massive dollar savings as well as significant emissions reductions across a fleet. On road tests are recommended.

Keywords: bus, aerodynamics; drag; fuel consumption; vehicle aerodynamics.

1. Introduction

As developing countries become more industrialised, their demand for petroleum has increased. At the same time petroleum supplies are known to be dwindling and this has led to large fuel price rises making all forms of transport expensive. The need for profitability requires the vehicle manufacturers to take advantage of all means of reducing fuel consumption.

The reduction of aerodynamic drag is one such area where fuel consumption can be decreased. Most transport vehicles such as trucks and buses are bluff bodies, thus having relatively large aerodynamic drag. This is principally made of pressure drag, of which form drag may be significant as well due to the presence of separation bubbles and vortex shedding. Due to high drag, buses and trucks have poor fuel consumption.

The primary aim of this study was to investigate aero-shaping means to reduce aerodynamic drag of bluff bodies, namely that of intercity buses. Aerodynamic drag reduction will have benefits in two areas; firstly, it will increase the profitability of the operator through lower fuel costs and

* Corresponding Author, E-mail: r.sharma@auckland.ac.nz

secondly it will reduce the consumption of a scarce environmental resource, fuel. Consequently, the burden on the environment will be lessened as well through reduced emissions.

Intercity buses provide long distance travel for passengers and their cargo between towns and cities. Due to the nature of their tasks, they travel at high velocities with relatively few stops meaning they are almost constantly operating at levels of high aerodynamic drag. As aerodynamic drag is proportional to the square of the bus velocity and the power required to overcome this drag is proportional to the cube of the bus velocity, any reduction in velocity or drag would lead to substantial fuel savings. An intercity bus cannot simply reduce its overall velocity because this would greatly increase the passenger travel time. Therefore the only option available to reduce the fuel consumption is to employ aerodynamic methods to reduce the overall drag on the bus. All modifications to the bus will need to provide financial benefit to the operator and not interfere with the buses primary function of transporting passengers. They will need to maintain internal passenger space and comfort, as well as being practical and achievable.

Recent research into drag reduction of bluff-bodied vehicles has focussed on trucks, buses, coaches and trains. A good example is SAE 850288 (1985), which discusses front-edge rounding and rear-edge shaping extensively. Many buses uses tapered tops and sides in conjunction with shaped rear ends and rounded front edges. This study will be able to compare past research on bluff bodies and will allow for the leading edge sides as well as the top to be altered to reduce drag. This can be done without overly affecting the internal passenger space, as the area at the front of the intercity bus is used for access and the drivers only.

2. Literature survey

Fig. 1 illustrates flow separation regions around a bluff-body such as a bus. The dynamic character of the flow field around a bluff body generally manifests in the form of flow stagnation in the front,

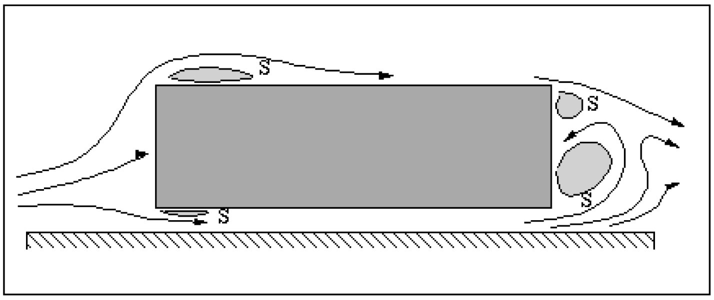


Fig. 1 Separation regions around a bus, side view (Fillipone 2003)

Table 1 Drag levels for large buses

Sections	Drag Percentage
Skin Friction	7
Fore Body	25
Under Body	30
Base	38

shear layer separation and reattachment on the sides, as well as turbulent-vortex wake downstream. In contrast to streamlined objects, bluff bodies have almost no pressure recovery region on the rear half of the body, resulting in high drag. Filippone (2003) summarises the total aerodynamic drag on large buses to be made up of skin friction, fore, under body and base drag, as shown in Table 1. Clearly, fore body and base drag make up 60% of the total drag, thus most aerodynamic drag reduction research is directed in these areas.

Drag reduction by aerodynamic means can be achieved through two methods; these are Active Boundary Layer Control (ABLC) and Passive Boundary Layer Control (PBLC). Both aim at reducing the frictional shear stresses and preventing or delaying the shear layer separation. ABLC is the more effective means of boundary layer control and are more versatile, but their application is energy demanding as they require power to operate and this power requirement must be removed from the drag reduction gains. PBLC is the control of the boundary layer through passive means such as aerodynamic shaping to reduce sharp changes in form, which produce stagnation points and separation lines. Methods of ABLC include the suction of low pressure air from the separation regions, the blowing of air into the boundary layer to re-energise the slower air near the surface and rotating surfaces to also re-energise the boundary layer. ABLC is more versatile, as it can be applied easily on any surface with the exception of the rotating cylinders, which require a certain amount of internal room to operate.

Munshi and Modi (1997) have shown that an average 36% reduction in drag can be achieved by using rotating cylinders at the forward leading edges of a bluff body. This Moving Surface Boundary Layer Control (MSBLC) idea, borrowed from high performance military aircraft technology, works by imparting momentum to the airflow delaying the boundary layer separation and virtually eliminating all vortex resonance.

Suction was first conceived by Prandtl in 1922 as a method of control and today it is often used in conjunction with blowing. It has been found that pure suction and combined blowing can be used in a positive manner to affect the vortex shedding, separation behaviour and the force coefficients characteristics.

Englar (2000) found that by blowing air out of the base of a truck and trailer unit, it was possible to improve the performance, stability and safety of the vehicle. It was found that by controlling the location of the blown air, either from the top or the side jets, it was possible to increase or decrease the lift and control the yaw experienced by side winds on the trailer. The blown air would reenergise the airflow in that region resulting in less pressure drag on that particular side of the vehicle, thus it provided a means of controlling the base of the trailer. A major flaw in this work was that the pumping energy required for blowing the jet was never accounted for. The harder the air was pumped, the higher the jet momentum was, and the lower the drag. Accounting for the pumping energy would inevitably negate any benefits.

Momentum injection using a rotating cylindrical element was shown by Kubo, *et al.* (1999) to be quite successful in suppressing the separation of the shear layers at the top and bottom faces of bluff bodies. It was found to provide control of flow separation from the leading edge through momentum injection. By altering the velocity of the cylinder the separation point could be controlled.

ABLC methods might all be considered impractical as no bus operator wants more mechanical complexity. PBLC methods and body shaping in particular, is therefore the primary acceptable method of drag reduction.

Methods of PBLC to date on large intercity buses include, the addition of aerodynamic spoilers to the rear of the bus. Through CFD modelling, Kim, *et al.* (2002) have shown the spoiler to be a successful add on device, which cleared away the extreme vortex produced at the upper body, and decreased aerodynamic drag and improved driving stability. Fletcher and Stewart (1986) placed a cavity at the base of a bus to generate a vortex which would turn the air flow towards the bus wake centreline and therefore reduce the size of the low pressure region and the result was that a full scale total drag reduction of 29% was achievable.

Prasad and Williamson (1997) have shown that by placing an object, such as a small wall upstream of a bluff body, a low pressure region is created behind the wall in which the bluff body is located. This produced drag reductions of up to 62% for the entire system consisting of the wall and bluff body. Such a device would be impractical on the open road and especially in built up areas like towns. Whereas a wall upstream of a bluff body can be considering as boundary layer tripping on a large scale, in that it fully encloses the bluff body. Smaller trip fences can be applied to the front flat surfaces of a vehicle to create local boundary layer tripping which reduces drag. When used on cube-like vans, the fence was found to have reduced drag by up to 31.4%; Modi, *et al.* (1995).

Camara and Girardi (1995) have shown that the rounding of the top leading edge of a truck model decreases the separation bubble dimensions in this frontal region and reduces overall drag by 40%. Cooper (2002) also studied a variety of PBLC methods on trucks with varying levels of success. The methods employed were cab shaping, cab mounted deflectors, trailer front end fairing, cab side extenders, body front edge rounding, tractor trailer gap seal, trailer side skirts and rear boat tails.

A boat tail is the construction of an enclosure on the base of the bus. Instead of the bus simply finishing as an abrupt cube shape, an enclosure that resembles the prism is built. An in-depth study into boat tails was made at the NASA Dryden Flight Research Centre (Meyer and Saltzman 1999) who determined that a drag coefficient of 0.242 was achievable for bluff body vehicles fitted with such a device. A full boat tail would be long and therefore impractical due to the dangers it presents and the study suggested the use of truncated boat tails.

Gao, *et al.* (1995) have shown that the vertical tapering of the vehicle windscreen at 18^{o} or greater was sufficient to eliminate the separation bubble at the top leading edge, thus reducing a major source of aerodynamic drag in bluff body vehicles.

3. Experimental details

3.1. The model

The experimental part of the study was based around a 1/20th scale model of a Mercedes Benz Tourismo 15 RHD bus, as shown in Fig. 2(a), that was manufactured for an earlier study of Lee (2003) and Oram (2003). The Tourismo as shown in Fig. 2b, is conventional, modern and has been manufactured in large numbers (Mercedes Benz Website 2004) and as such is a good representative of an intercity bus. It also complies with the New Zealand Land Transport Safety Authority guidelines on the dimensions of large buses (LTSA 2004(a)).

The scale model was constructed of MDF sheets; the sheets had been laminated together and machined to the final shape. The model has a detachable face and base as shown in Fig. 2(a), which allowed different aerodynamically shaped combinations to be tested and compared. The model also has wheel arches, axles and wheels. For simplicity the model does not have rear view mirrors, engine or accessory inlets.

Fig. 2 1/20 scale model (left) of the Mercedes Benz Tourismo 15 RHD bus (shown on right)

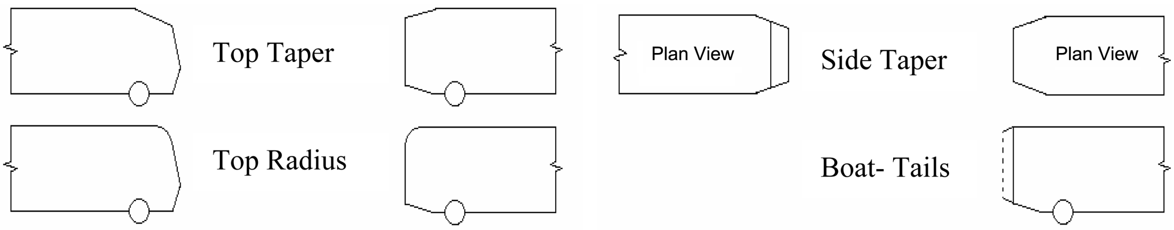


Fig. 3 The modifications that were tested individually, and then in combination

3.2. Ideas tested

A detailed study of modern intercity buses showed that they are becoming higher, providing room underneath for paying cargo and in some cases a second level of passengers. In doing so a large amount of unusable internal volume is created at the front of the bus. In fact the front meter and a half of many buses is solely dedicated to the drivers position and to access for the passengers. This space could easily be used for aerodynamic shaping of the bus without affecting any paying passenger seats, thereby not affecting revenue.

It was thus deemed practical and thus possible to lower the top leading edge by curving or tapering without affecting the field of view of the driver or the passengers. The sides of the bus could also be tapered in. To deal with base drag, the same tapering could be applied to the rear edges but this will definitely affect passenger paying seats area as the last row would be lost, although many buses now use this area for floor to roof baggage storage. As many buses in New Zealand are shorter than the 12.6 m length required (LTSA 2004a), it is possible to add boat-tails to the base to turn the air towards the centre of the bus reducing the overall size of the low pressure region. The modifications that were tested are shown in Fig. 3. In addition, two combinations were also tested. All tapers including the boat-tail were set at 18^o, which was based upon initial CFD analysis. For his bluff bodied vehicles and trucks, Cooper (1985) had found most benefits in the vicinity of 15^o.

3.3. Wind tunnel testing

All wind tunnel testing was carried out at the University of Auckland, School of Engineering in

Fig. 4 The model mounted in the wind tunnel on an elevated ground plane. A mechanical force balance is visible underneath the test section

the high speed section of the de Bray wind tunnel, which has a 600 mm by 800 mm square test section. It is equipped with a three-component (drag force, lift force, pitching moment) mechanical force balance, to which the model is mounted via rigid support rods; see Fig. 4. Forces on these rods were measured prior to actual model testing and accounted for in the final data analysis. Speeds of up to 60 m/s (216 kph) can be generated in the tunnel. Pressure probes connected to an electronic manometer enables the air velocity to be measured. Mounting brackets are also provided for a raised floor to reduce the boundary layer effect on the model, as can be seen in Fig. 4. Calculations based on flat plate correlations at 100 kph showed that the boundary layer thickness on the ground plane was around 22 mm at the rear of the bus.

3.4. Similarity

To maintain similarity, it would be ideal to match the Reynolds numbers at model and full-scale. However since the model is 1/20th scale, this would require wind tunnel velocities of 20 times the full-scale velocity, which is not practical. The coefficient of skin friction drag is inversely proportional to Reynolds number so the model will have a slightly higher skin friction drag coefficient than the full sized bus. For bluff bodies, the skin friction drag is small compared to pressure drag. The point of flow separation for a sharp edged bluff body and hence the fore-body drag coefficient is virtually independent of Reynolds number, as is the base drag coefficient. However for rounded edges, the fore-body drag would usually be sensitive to Reynolds number if flow separation is present, since the separation points on curved or rounded surfaces are Reynolds number dependent. Hucho, *et al.* (1976) show that lack of Reynolds number similarity is likely to produce higher drag coefficients at model scale as compared with full scale for the same curvatures. However, elimination of separation is one of the main objectives of this study. Consequently the overall drag coefficient should not be greatly affected by the failure to match the model to the full scale Reynolds number. Notwithstanding this, Reynolds number sensitivity was nevertheless investigated in this study and tests were carried out at 80, 90 and 100 kph. Even though the Reynolds numbers here (i.e. in the wind tunnel) were all in the subcritical range, the CFD analysis considered the full scale situation, and the assumption of insensitivity of the flow and the drag to Reynolds number was able to be confirmed.

3.5. Blockage

The dimensions of the bus relative to the tunnel correspond to a 4.5% blockage ratio. Wind tunnel blockage ratios of 5% or less are considered low enough to allow valid results to be obtained (Perzon, *et al.* 1999). Since the model had a blockage level very close to the acceptable limit, corrections were made for blockage in the calculation of drag coefficients obtained from the wind tunnel. The correction method used was utilised by Carr, *et al.* (1986) to account for blockage in wind tunnel tests on model trucks. The corrected drag coefficient (C_D) was calculated using:

$$C_D = (C_{Du} + \Delta C_{Db} \cos \psi)(1 - 2A/A_2) \tag{1}$$

where

$$C_{Db} = \frac{-2V}{(2A_2)^{3/2} C_{Duw}} \tag{2}$$

and

A = Model frontal area (m^2)
A_2 = Cross-sectional area of the wind tunnel above the ground plane
C_D = Drag coefficient corrected for blockage
D_{Du} = Uncorrected drag coefficient in body axis coordinates
D_{Duw} = Uncorrected drag coefficient in wind axis coordinates
V = Model volume (m^3)
ΔC_{Db} = Drag increment due to wake buoyancy
ψ = Yaw angle

4. CFD modelling

The commercial software package CFX^{TM} version 5.7 was used for computational fluid dynamics (CFD) modelling of the bus and its modifications. The main object of using CFD modelling was to complement wind tunnel experiments with flow visualisation, in order to be able to understand and explain the results better.

A major problem in CFD is the accurate modelling of flow separation from a surface, which is strongly dependent on the turbulence model. The two equation k-ε model finds widespread use due to its robustness and ability to provide turbulent flow predictions with reasonable accuracy. It is however not recommended for transport modelling as it fails to detail the onset of separation and the amount of separation under adverse pressure gradients (Kral 1998). The Shear Stress Transport

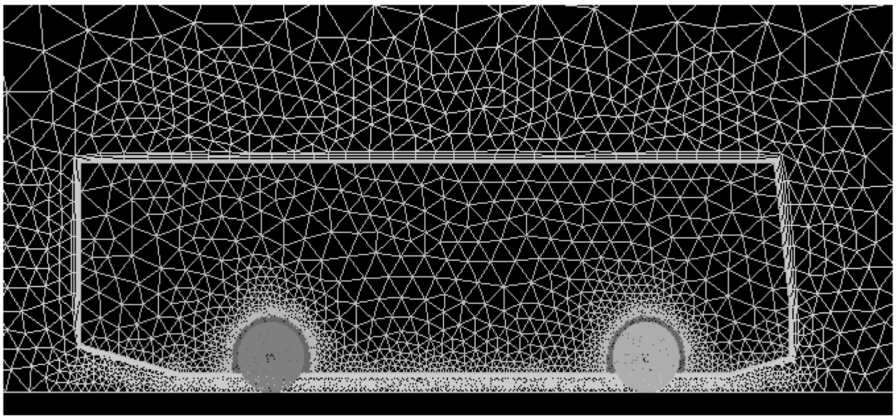


Fig. 5 Mesh near the bus (height H). (The domain is 12H upstream, 12H downstream and 10H verti-cally)

(SST) model of Menter (Kral 1998) includes a viscosity limiter that recreates larger separation bubbles that are more representative of wind tunnel experiments. The SST model is a blended k-ε / k-ω model and has been shown in studies to greatly improve the prediction of turbulence in a transport model. The SST turbulence model was thus utilised in this study.

To ensure the correct modelling of the pressure stagnation point in front of the model, it is necessary to have the domain inlet situated well forward of the model. (Perzon, *et al.* 1999) recommend that the inlet be more than 10 model heights upstream of the model. The outlet at the rear of the domain also needs to be taken into account due to the low pressure wake region at the base of the model, and should be at least 10 – 12 model heights downstream. The computational domain used for the simulations in the present study was basically half a vehicle to the symmetrical nature of the airflow over the vehicle. To ensure that the domain boundaries did not influence the near body flow, the inlet was placed 12H (H = Height of bus) in front of bus, the outlet 12H behind the bus and the top of the domain was 10H above the bus while the sidewall was 5W (W = Width of Bus) away from the bus.

The geometry for the model and the computational domain was created using ProEngineer™ and imported into the CFX™ associated ICEM4™ meshing package. The meshing algorithm allows the user to apply a larger 'Global Mesh Size' to the domain and a finer 'Surface Mesh Size' mesh to selected surfaces. An automatic mesh size reduction option called 'Natural Mesh Size' varies the mesh size down to the minimum mesh size stated by the user. A mesh refinement study was conducted with the original bus model. As the drag forces calculated by the package was shown to have become apparently insensitive to further refinement, a mesh similar to that illustrated in Fig. 5 was utilised, which only shows the mesh near the bus. The total number of mesh cells was close to 2×10^6.

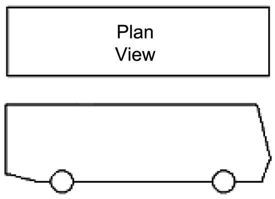
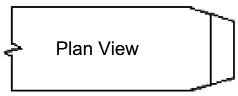
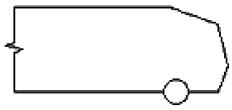
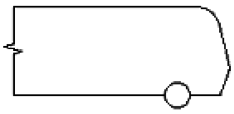
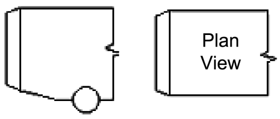
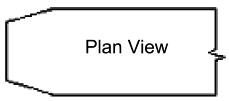
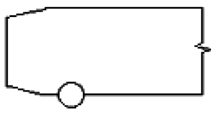
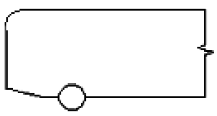
5. Results and discussions

5.1. Wind tunnel tests: original bus model and individual modifications

The drag coefficient measured in the wind tunnel for the original bus model was $C_D = 0.86$, which lies in the range 0.5 – 0.85 (Englar 2000) and 0.80 – 0.90 (McGuiness 2004/2005) found in the published literature. This provides confidence as to the adequacy of the wind tunnel modelling employed in this study. As a further comparison, the measured drag coefficient is also reasonably close to the value of 0.80 obtained by Lee (2003) and Oram (2003) in their tests of the same model in the same wind tunnel in an earlier study. The discrepancy between the two values may be attributed to the fact that Lee and Oram used a simplified method to account for blockage.

Table 2 summarises the drag (C_D) and lift (C_L) coefficients measured in the wind tunnel at 90 kph,

Table 2 Aerodynamic forces measured in the wind tunnel at 90 kph for the original (control) bus model and for the modified models

Model		Force Coefficients Based on Measured Forces		% Change Compared to the Control Bus	
Description	Schematic	Drag	Lift	Drag	Lift
Control Bus Un-Modified		0.859	-0.084	N/A	N/A
Fore-body with 18° Tapers on the side Edges		0.479	-0.065	-44	23
Fore-body with an 18° Taper on the Top Edge		0.647	-0.062	-25	26
Fore-body with a 25 mm Radius on the Top Edge		0.690	-0.062	-20	25
Tail with External Boat-tails Attached (18° Taper)		0.780	-0.033	-9	60
Tail with 18° Tapers on the Side Edges		0.795	-0.026	-7	69
Tail with an 18° Taper on the Top Edge		0.806	-0.023	-6	73
Tail with a 25 mm Radius on the Top Edge		0.836	-0.053	-3	37

for the original or control bus model, and for individual modifications applied to it.

The intended effect of the fore-body modifications was to reduce separation at the leading edges, thereby reducing the fore-body component of vehicle drag. Table 2 shows that both tapering and rounding of the top leading edge resulted in significant drag reductions of 25% and 20% respectively. The likely explanation is that both methods significantly reduce or eliminate flow separation. Furthermore, tapering of the side leading edges results in dramatic drag reductions of 44%, which is approximately twice that for a tapered top leading edge. Considering that side leading edge tapering eliminates or minimises two separated flow regions, one on either side of the front of the bus, this result is therefore not surprising. These effects will become clearer when the CFD flow visualisation results are presented and discussed in the next sub-section. There is a significant amount of free internal space at the front of the bus as it is solely occupied by the driver. Consequently there is scope for tapering the side leading edges without compromising function or passenger carrying capacity. Also, as there is significant headroom above the driver, there is scope for either rounding or tapering the top leading edge.

The intended effect of the tail modifications was to assist the flow in turning towards the bus centreline. The resulting turbulent wake area would be shorter which would enhance the pressure recovery on the back surface of the bus, reducing the base drag component. Both rounding and tapering of the trailing edges appear to reduce base drag by a small amount. As found for the fore-body modifications, tapering appears to be more effective than rounding. The drag reduction of 7% measured for the tapered trailing edge is significantly less than the drag reduction of 20% measured by Fletcher and Stewart (1986). This discrepancy is due to differences in the test procedure, bus body geometry and more importantly in taper geometry. The drawback of tapering the trailing edges is that passenger-carrying capacity will be reduced. The side tapers will reduce the back seat capacity by atleast two passengers while the top taper may eliminate the entire back row as there is little head room at the rear of the bus. The boat-tails provide tapering on all four trailing edges and they appear to be the most effective in reducing base drag. The measured drag reduction of 9% compares reasonably well with the reduction of 5% measured by Cooper (2003) during tests on a truck. Since the boat-tails are external, they do not compromise the passenger space in the rear of the bus. The boat-tail dimensions on the model were restricted to ensure com-pliance with regulations regarding maximum length of buses.

5.2. Wind tunnel tests: combined modifications

The results obtained from tests on the original bus model and individual modifications discussed above show that the leading edge tapering and rounding at the front of the bus, and a boat tail at the rear are the most effective modifications in terms of drag reduction. Based upon this, it was decided that two combination models would be investigated. Both include the boat-tails at the rear as they were the most effective at reducing drag in this region while sacrificing no internal space. Two fore body sections were considered, both with side tapers. The first of these included a tapered top leading edge, while the second with a rounded top edge and curved front and immediate tapered sides. The two combination models referred to as Combo 1 and Combo 2 respectively are shown in Fig. 6.

Since Combo 2 has a more rounded fore body with large radiuses, it was decided that Reynolds number sensitivity, if any, be studied and additional tests were thus conducted at 60, 80 and 100 kph besides the standard test at 90 kph. CFD modelling was conducted for the models as well.

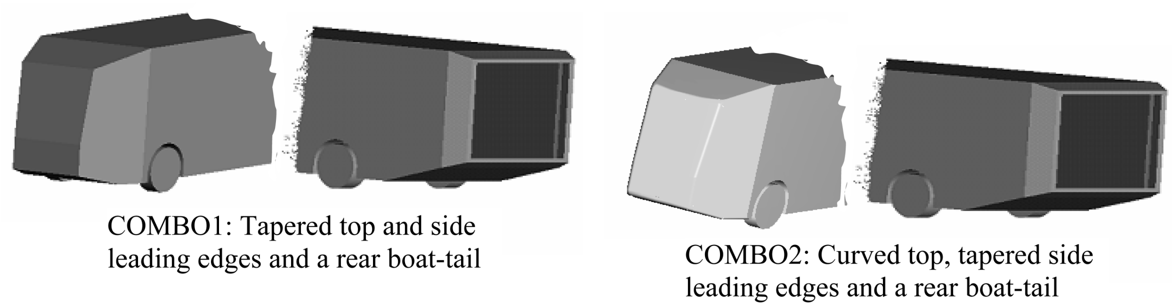


Fig. 6 Combination models

Table 3 Drag and lift coefficients measured in the wind tunnel at 90 kph for the original (control) bus model and for the combinations models

	Control model	Combo 1	Combo 2
C_D	0.86	0.29	0.25%
Change in C_D	-	-65.8	-70.9
C_L	-0.084	0.003	0.011

The results from the wind tunnel testing are presented in Table 3, and which show that the modifications have dramatically reduced the overall drag coefficient C_D by up to 65% for combination 1 and 70% for Combination 2 respectively. Further tests conducted for Combination 2 model, showed no Reynolds number sensitivity.

The decreases in the drag coefficient are accompanied by increases in the lift coefficients C_L, as shown in Table 3. The estimated maximum speed that an intercity bus is likely to be driven at is 100 kph (10 kph over the maximum legal speed of 90 kph for vehicles over 3500 kg (LTSA 2004(b)). At this speed, the corresponding increase in lift forces for the Combo 1 and Combo 2 models extrapolated to full scale is 120 kg and 133 kg. The increase in lift equates to less than the combined weight of two passengers with an estimated average weight of 70 kg each. In the context of a bus such as the Mercedes Tourismo which carries up to 49 passengers, a driver and luggage, this does not seem to be a significant increase in lift. However, without a detailed knowledge of the handling characteristics of the Mercedes Tourismo it is difficult to evaluate the exact conse-quences of this increase in lift.

5.3. CFD modelling: validation

CFD analysis of the Control, the Top Taper, Top Radius and Combinations 1 and 2 was carried out to provide information concerning the stagnation, separation, reattachment and wake conditions associated with the models. To validate the visual results, the drag coefficients C_D were also calculated in the CFD simulation and compared with the corresponding data obtained from wind tunnel testing. These comparisons appear in Fig. 7, and which show that there is remarkable agreement of C_D values between the wind tunnel and the CFD simulation results, both in terms of the trend, as well as in terms of the absolute coefficients. This was taken as validation that the CFD modelling was in fact reasonable and that the visualisations produced by it could be used with confidence.

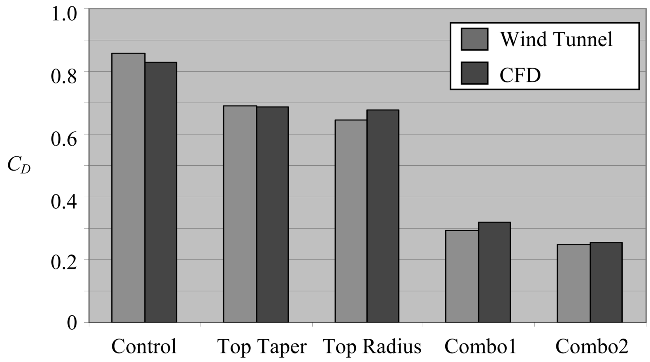


Fig. 7 Comparison between wind tunnel and CFD results

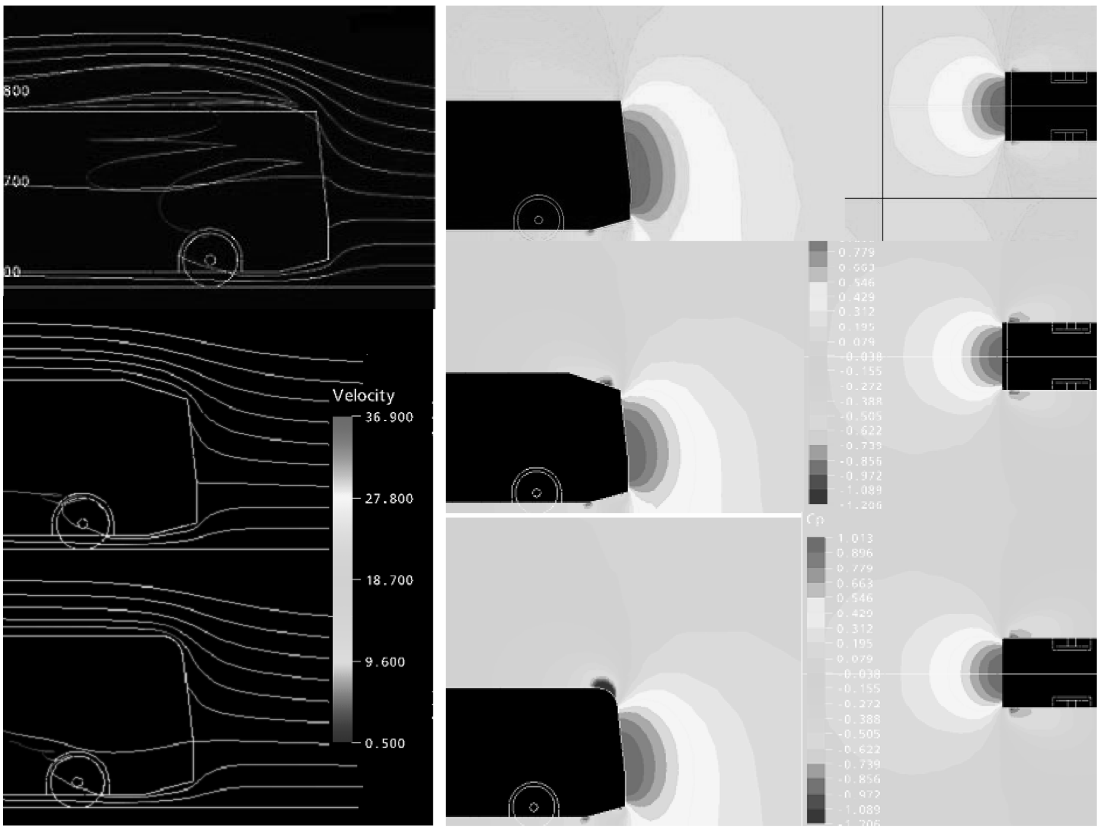


Fig. 8 Streamlines and pressure distributions obtained from CFD for the original bus model, and for fore-body top taper and top radius modifications

5.4. CFD modelling: fore-body modifications

In wind tunnel tests, it was found that both leading edge top radius and the top taper modifications produced significant reductions in drag. To examine and understand this further, CFD flow visualisation was conducted using the results of CFD modelling. Streamline and pressure distribution plots thus obtained for these two cases are compared in Fig. 8, which highlight the following reasons for the reduction in drag coefficient measured for both the tapered top edge and rounded top edge models:

1. There is strong flow separation at the top leading edge of the control bus, as expected. The size of this bubble is significantly reduced if not eliminated by both tapering and rounding. This indi-cates that flow separation and hence fore-body drag has been reduced in both cases.
2. The size of the high pressure stagnation area in front of the bus is reduced both by tapering and by rounding the top leading edge. This is further accompanied by a frontal region near the top where the pressure is actually negative hence further negating the drag. The latter appears to be slightly more so for the top taper, hence a $C_D = 0.65$ was recorded in the wind tunnel, as compared with a $C_D = 0.69$ for the case of the top radius.

The measured drag reduction of 20% due to rounding the leading edge is reasonably close to the value of 27% measured by Gao, *et al.* (1995), who used a different bus model.

5.5. CFD modelling: Combo 1 and Combo 2 models

The coefficient of pressure plots in Fig. 9 show that Combo 1 and Combo 2 both have the effect of reducing the size of the positive pressure area in front of the bus. For Combo 2 this area is slightly smaller than for Combo 1. This indicates that even the slight curvature of the front surface

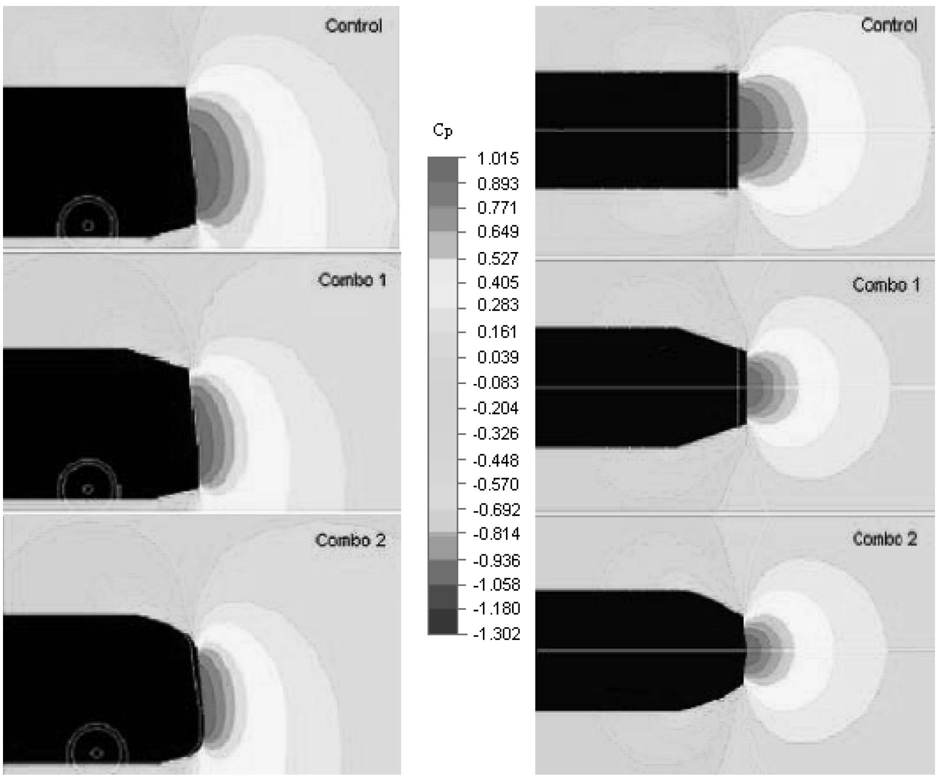


Fig. 9 Pressure distributions obtained from CFD for the original and modified bus models

of the Combo 2 model brings about a further reduction in base drag. The streamline plots of Fig. 10 show quite clearly, the dramatic elimination of the leading edge separation regions in the modified models, as compared with the situation for the original or control model. That significant reduction in drag can be achieved as already shown from wind tunnel tests, is not surprising from these plots.

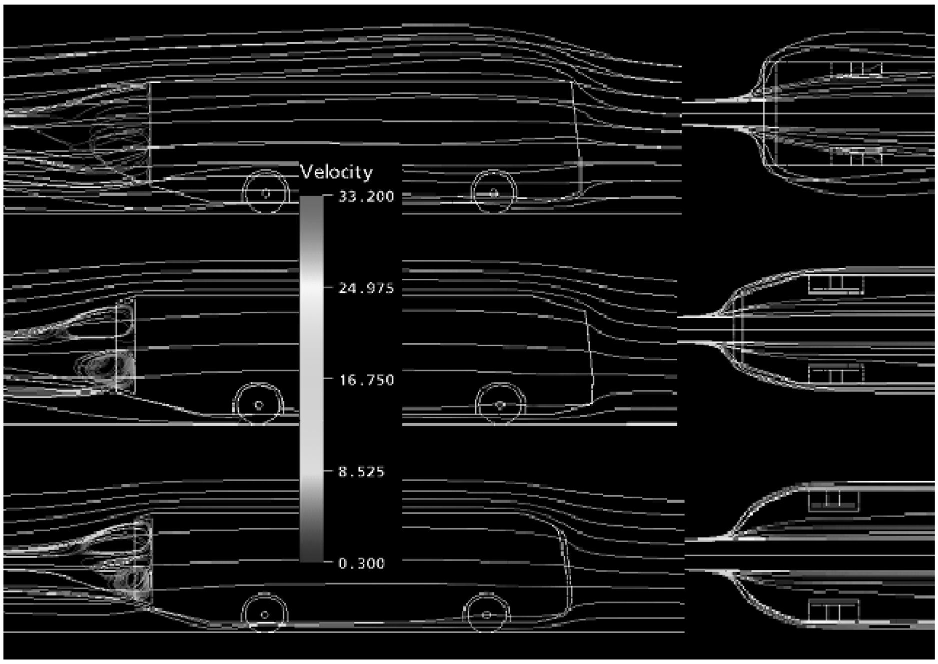


Fig. 10 Streamlines obtained from CFD for the original and modified bus models

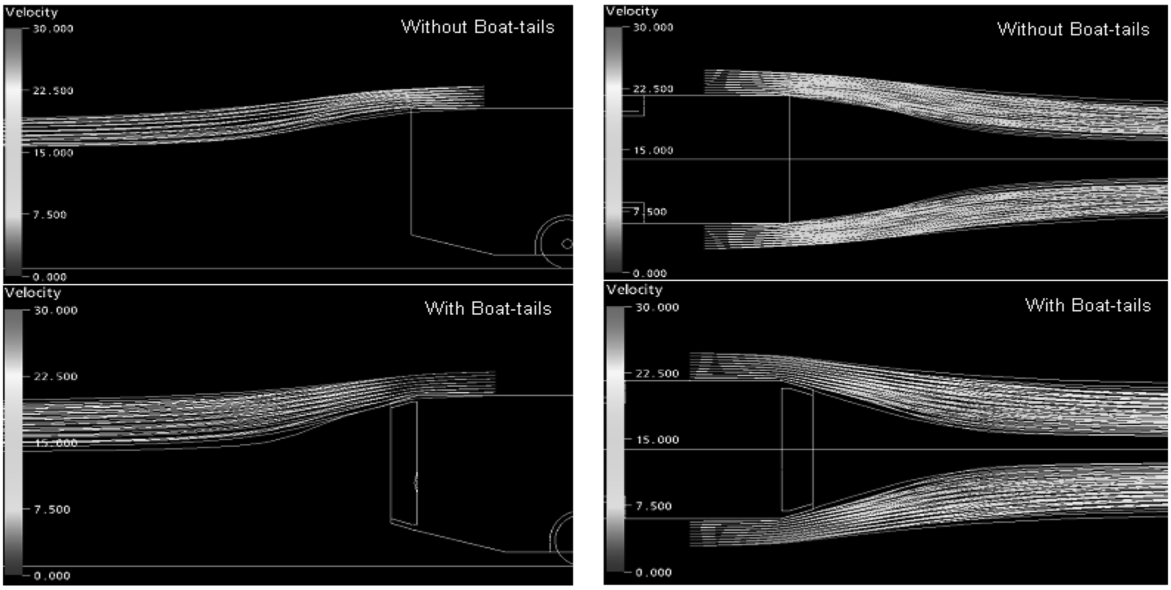
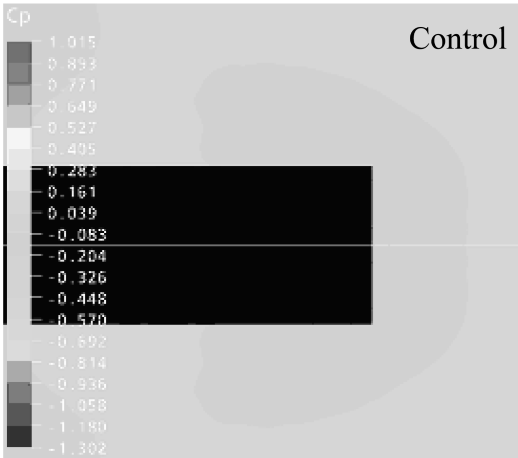


Fig. 11 Streamlines from CFD showing the flow around the rear boat tail

The streamline plots in Fig. 11 show that boat-tails have the effect of causing the flow along the top edge to turn more sharply towards the bus centreline. The flow along the bottom is not greatly affected by the boat-tails due to the fact that the under-body of the control bus is already sloped at a similar angle to the boat-tails. The effect of the altered wake flow can be seen in the rear pressure plots in Fig. 12. It shows that the rear pressure increases (i.e. becomes less negative) due to improved pressure recovery at the back of the bus associated with the boat tail. Thus the pressure differential between the front and back of the bus is reduced and therefore base drag is further reduced by the addition of the boat-tails.

6. Discussions on fuel consumption

Fig. 13 shows the breakdown of the components of fuel consumption for a heavy vehicle such as a bus travelling at 90 kph along a level road. Aerodynamic drag contributes to 40% of fuel consumption. Based on this, a 70% reduction in aerodynamic drag as measured for the Combo 2 model corresponds to a fuel savings of 28% for a typical bus on intercity or interstate operation on



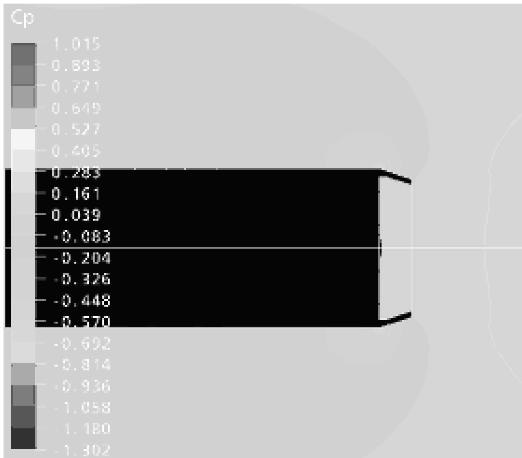


Fig. 12 Pressure plots from CFD showing the pressure distribution at the rear of the bus models

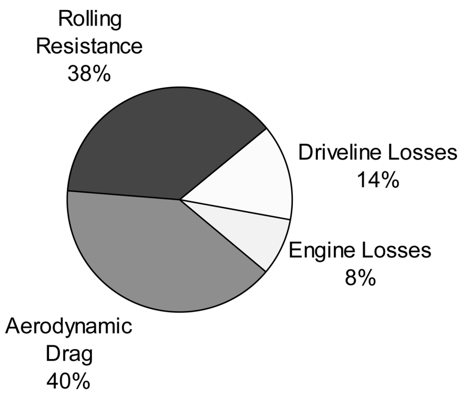


Fig. 13 Breakdown of components of fuel consumption for a heavy vehicle at 90 kph

mostly level roads. This value will however vary according to other factors such as vehicle weight and road incline, but nevertheless is quite a significant reduction. It could translate to a massive dollar savings as well as significant emissions reductions across a fleet.

7. Conclusions

A number of passive aerodynamic drag reduction methods were applied separately and then in two combinations on an intercity bus model, through wind tunnel studies on a 1:20 scale model of a Mercedes Benz Tourismo 15 RHD intercity bus, and computational fluid dynamics (CFD) modelling. CFD flow visualization reveals that the curved front section and tapered top and side edges eliminate flow separation in these areas and reduces the size of the positive pressure area at the front of the bus. The use of a boat-tail at the rear of the bus helps reduce the size of the low pressure region at the rear. Furthermore, it was found that the magnitude and trends in the force coefficients obtained from CFD modelling exhibit remarkable agreement with experimental results. From the results obtained, it is concluded that up to 70% reduction in aerodynamic drag (C_D) can be achieved on the model using tapered and rounded top and side leading edges, and a truncated rear boat-tail. An analysis shows that such a reduction in aerodynamic drag could lead to a significant 28% reduction in fuel consumption for a typical bus on intercity or interstate operation. This could translate to a massive dollar savings as well as significant emissions reductions across a fleet. The aerodynamic shaping in the fore body of the bus is quite practical, as there is sufficient room in the front sides and top for tapering or rounding to be applied, without compromising passenger space. On road tests are recommended.

Acknowledgements

The original 1:20 scale bus model upon which this study was based was designed for an earlier study by C J Lee and W J Oram under the guidance of Professor Richard Flay. Their permission to utilise the model is gratefully acknowledged.

References

Camara, E. and Girardi, R. (1995), “On the aerodynamic analysis of a bus model”, *Society of Automotive Engineers Brazil*, 952265E.

Carr, G. W., Cooper, K. R., Garry, K. P., Gerhardt, H. J. and Whitbread, R. (1986), “A comparison of aerodynamic drag measurements on model trucks in closed-jet and open-jet wind tun-nels”, *J. Wind Eng. Ind. Aerodyn.,* **22**, 299-316.

Cooper, K. R. (1985), “The effect of front-edge rounding and rear-edge shaping on the aerodynamic drag of bluff vehicles in ground proximity”, *Society of Automotive Engineers*, Technical Paper SAE850288.

Cooper, K. R. (2003), “Truck aerodynamics reborn-lessons from the past”, *Society of Automotive Engineers*, 2003-01-3376.

Englar, R. J. (2000), “Development of pneumatic aerodynamic devices to improve the perform ance, economics, and safety of heavy vehicles”, *Society of Automotive Engineers*, 2000-01-2208.

Filippone, Dr A. (2003), *Advanced Topics in Aerodynamics*, Retrieved June 2004, from: http://aerodyn.org/

Fletcher, C. A. J and Stewart, G. D. H. (1986), “Bus drag reduction by the trapped vortex concept for a single bus and two buses in tandem”, *J. Wind Eng. Ind. Aerodyn.*, **24**, 143-168.

Gao, Li. Yinsan, Chen. Gong, Chunyuan. and Gao, Yanling. (1995), “Automobile model scale wind-tunnel test by means of aeronautical wind-tunnel”, *Society of Automotive Engineers*, 950999.

Hucho, W. H., Janssen, L. J. and Emmelmann, S. J. H. (1976), "The optimisation of body details-a method of reducing aerodynamic drag of road vehicles", *Society of Automotive Engineers*, 760185.

Kim, M., Kuk. J. and Chyum, I. (2002), "A numericial simulation on the drag reduction of large sized bus using rear spoiler", *Society of Automotive Engineers*, 2002-01-3070.

Kral, L. D. (1998), "Recent experience with different turbulence models applied to the calculation of flow over aircraft components", *Progress in Aerospace Science*, **34**, 481-541.

Kubo, Y., Yukoku, E., Modi, V. J., Yamaguchi, E., Kato, K. and Kawamura, S. (1999), "Control of flow separation from leading edge of a shallow rectangular cylinder through momentum in-jection", *J. Wind Eng. Ind. Aerodyn.*, **83**, 503-514.

Lee, C. J. (2003), *Aerodynamic Drag Reduction of Buses*, Mechanical Engineering Project, School of Engineering, The University of Auckland.

LTSA. (2004a), *Fact Sheet 11*, Retrieved March 2004, from www.ltsa.govt.nz.factsheets/11.html

LTSA. (2004b), *Amendment Regulations-Heavy Vehicles Speed Limits*, Retrieved September 21, 2004 from: http://www.ltsa.govt.nz/legislation/hvy-veh-speed-limits-qa.html

McGuinness, H. J. (2004/2005), *Air Resistance: Distinguishing Between Laminar and Turbulent Flow*, Retrieved September 18, 2004, July 2005, from: http://academic.reed.edu/physics/courses/phys100/Lab20Manuals/Air20 Resistance/Air.Resistance.pdf

Mercedes Benz Omnibus Website, Retrieved June 2004, from: www.mercedes-benz.com/e/ecar/omnibus/default.htm

Meyer, R. and Saltzman, E. (1999), "A Reassessment of Heavy Duty Truck Aerodynamic Design Features and Priorities", *NASA Technical Publications*, TP-1999-206574, 1-33.

Modi, V. J., Hill, S. St, and Yokomizo, T. (1995), "Drag reduction of trucks through boundary-layer control", *J. Wind Eng. Ind. Aerodyn.*, **55/56**, 583-594.

Munshi, S. R. and Modi, V. J. (1997), "Aerodynamics and dynamics of rectangular prisms with moving momentum injection", *J. Fluids Struct.*, **11**, 873-892.

Oram, W. J. (2003), *Aerodynamic Drag Reduction of Buses*, Mechanical Engineering Project, School Of Engineering, the University of Auckland.

Perzon, S., Janson, J. and Hoglin, L. (1999), "On comparisons between CFD methods and wind tunnel tests on a bluff body", *Society of Automotive Engineers*, 1999-01-0805.

Prasad, A. and Williamson, C. H. K. (1997), "A method for the reduction of bluff body drag", *J. Wind Eng. Ind. Aerodyn.*, **69-71**, 155-167.

CC