15th Australasian Fluid Mechanics Conference
The University of Sydney, Sydney, Australia
13-17 December 2004

Drag Reduction on Bluff Bodies using a Rotating Device

S.E. Robson and N.A. Ahmed

School of Mechanical & Manufacturing Engineering
The University of New South Wales, Sydney, NSW, 2052 AUSTRALIA

Abstract

Drag plays a significant role in the overall force on aerodynamic bodies. In this paper a drag reducer in the form of a rotating ventilator is proposed for bluff bodies. The qualitative and quantitative investigations were conducted on the effects of this device mounted on the front end of a rectangular bluff body. Results indicate that the device decreases drag on the body significantly by approximately 50% at Reynolds numbers of 3.5 x 10^5 and 6.5 x 10^5. The incorporation of this device also makes the drag of the body less Reynolds number dependant. Consequently the proposed configuration is more efficient at higher speed in comparison to the conventional configuration which did not have the device attached to it.

Nomenclature

C_D	drag coefficient, $\dfrac{D}{\frac{1}{2}\rho V^2 S}$
D	drag force
L	characteristic length
Re	Reynolds number, $\dfrac{\rho V L}{\mu}$
S	reference area, defined as frontal area
V	free stream velocity
μ	free stream viscosity
ρ	free stream density

Introduction

Bluff bodies are used in a wide variety of applications such as bridges, buildings and other infrastructure and transport vehicles. The forces and characteristics of the flow around these objects can be significant for example vortex shedding and galloping may cause structural vibration problems while excessive wind loads may cause static structural problems and/or excessive energy consumption. Extensive studies have been conducted into how to reduce wind loads and vibration loads on bluff bodies. Boundary layer control devices and streamlining of the body are the two major methods of reducing loading, 80% of which may be attributed to pressure drag.

Although a variety of techniques are currently used to reduce drag and consequently fuel consumption or required structural strength many have a limited range of application. The most popular forms of aerodynamic improvements for bluff bodies use passive flow control techniques that effectively streamline the shape such as steps [8], flow deflectors [15], fences [8], [12], rounded edges [5], [8] and [15], nose cones [15] and wake ventilation [6]. However, these devices have a limited operating envelope.

Boundary layer control using rotating control cylinders, on the other hand, has shown greater promise on various shaped bodies [1][7],[9]-[11], [13]. However, the requirement of large control power input and other associated costs have not made the techniques commercially viable at present [7], [13]. The ideal solution to the problem of drag reduction should be competitive on cost and applicable in a large range of realistic flow situations.

If the main functional purpose of the bluff body, which may be to hold a certain capacity [buildings and other infrastructure] or allow the conveyance of goods [transport vehicles] or passage of people and vehicles [bridges], is not to be compromised, the current rectangular shape of these bodies appears to be appropriate. The drag characteristics of a rectangular box exhibit a direct relationship with the sharpness of the corner. Most of the drag is produced by losses induced at the corner. The more rounded the corner the lower the losses and the drag. However, most transport vehicles and infrastructure require the useable storage or road space to be maximised and tend to have sharp corners and edges. Active methods of drag reduction such as control cylinders must have a minimum energy input.

In this paper, the use of a generic rotating ventilator as a form of low energy drag reducer is proposed for use on bluff bodies such as transport vehicles. An experimental investigation of a rectangular bluff body was conducted to demonstrate the viability of this novel concept. The ventilator is expected to produce an effect on the flow characteristics similar to a rotating cylinder but with the added advantage that it can be operated using natural wind without the need for any power input by the engine or any auxiliary unit. Another advantage of this configuration is that storage capacity is unaffected as it is mounted externally.

Experimental Set Up

The bluff body was modelled as a rectangular box with dimensions of 0.3x0.3x0.5m. The ventilator was mounted on the front of the model with the axis of rotation parallel to the front surface and the bottom edge as depicted in Figure 1. A commercial wind driven ventilator, Edmonds model number GP130, was used in all experiments. A full description of the ventilator can be found in Rashid and Ahmed [14].

The 30" Open Circuit Open Jet Wind Tunnel in the Aerodynamics Laboratory at the University of New South Wales [2] was used to conduct the experiments. The test Reynolds numbers were selected to lie between 3.5 - 6.5 x 10^5 with the major study at an Re of 5.4 x 10^5 where the characteristic length is defined as the length of the bluff body. The model was mounted in the tunnel with provision for the angle of yaw to be adjusted. Measurements were taken at yaw angles of 0, 5, 10 and 15 degrees. The Reynolds number dependence of the zero angle case was examined at Re of 3.6 x 10^5, 4.5 x 10^5, 5.4 x 10^5 and 6.3 x 10^5.

Tufts of synthetic woven thread were used for flow visualisation. A six axis Industrial Automation ATI DAQ F/T Gamma Load Cell was used to measure forces on the model. The load cell was connected to a PCI-6034E National Instruments Data Acquisition Card. The data acquisition program provided by the manufacturer was used to convert the raw voltages into force output that was displayed on the monitor. The test section arrangement is shown in Figure 2.

Three configurations were tested. Configuration 1 was a plain box, Configuration 2 had the ventilator mounted on the box but rotation was prevented and Configuration 3 was the box with the ventilator free to rotate.

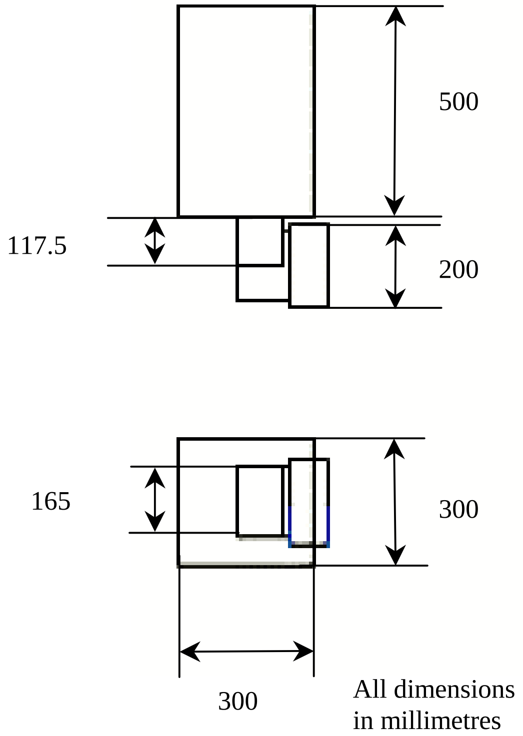


Figure 1. Bluff Body and Ventilator Model.

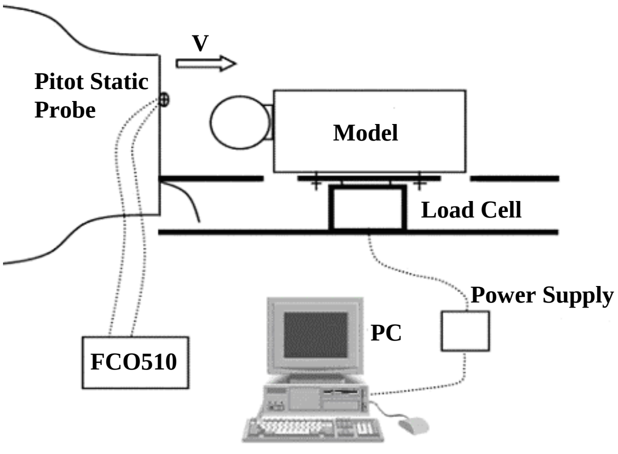


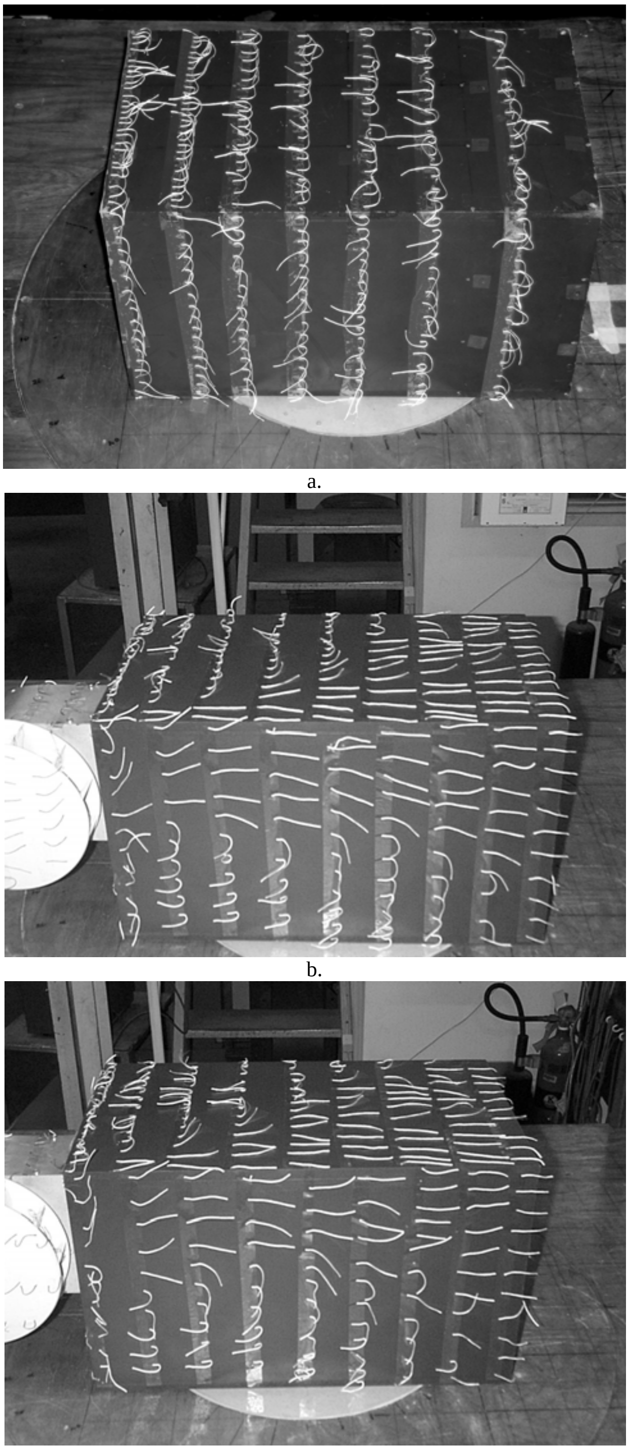
Figure 2. Test Section and Equipment Arrangement.

Results and Discussion

Qualitative Results

Results of the flow visualisation are shown in Figure 3. There is an obvious improvement in flow attachment of Configuration 2 over Configuration 1. The improvement in Configuration 3 over Configuration 2 is however, less pronounced. There is further improvement in Configuration 3 where in row five, seven straight tufts can be seen in this configuration as opposed to four straight tufts in Configuration 2. The increased straightening of the tufts is visible on most of the top surface and some of the side surface indicating improved streamlining and hence reduced drag of the body.

The flow visualisation indicates results similar to those found for rotating control cylinders on bluff bodies, which draw the flow back down onto the body, reducing separation [1], [7], [10][11] [13].


a.

b.

c.

Figure 3. Flow Visualisation for Re = 5.4 x 10^5.
a. Configuration 1, b. Configuration 2, c. Configuration 3.

Quantitative Results

Results of the quantitative investigation are presented in terms of drag force coefficients. The drag coefficient results shown in Figures 4 and 5 indicate a significant improvement due to the ventilators rotation. Data for Configuration 1 suggests the drag coefficient results were between 0.92 and 0.8 at yaw angles of 0 to 15 degrees. To obtain confidence in the data obtained, a check was made on the value of the drag coefficient for Configuration 1 using ESDU data sheet 71016 [4]. The data sheet gives a value of one for a surface mounted block which is close to the value obtained in this experiment for this configuration and corresponds within the bounds of the published uncertainty.

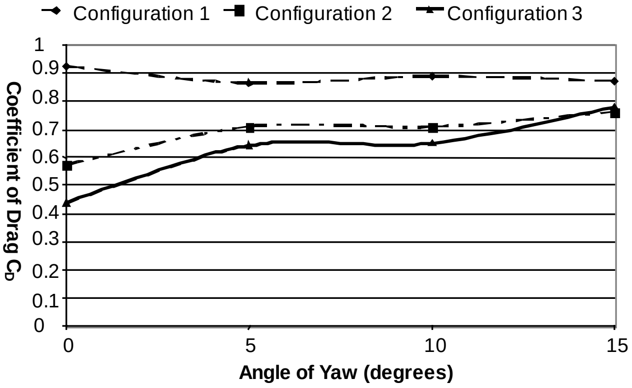


Figure 4. Variation of C_D at different angles of yaw for Re = 5.4 x 10^5.

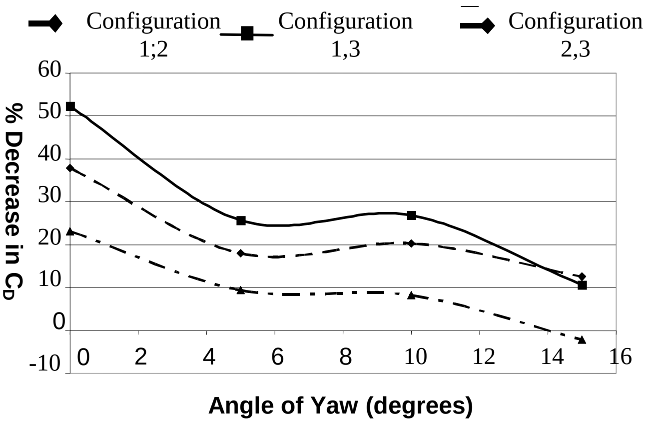


Figure 5. Percentage Decrease in Drag Coefficient for Re = 5.4 x 10^5.

The effect of the ventilator is found to be most significant at 0 degrees angle of yaw with a decrease in drag coefficient of over 50%. At higher angles of yaw the improvement in drag coefficient is lower but is still significant. In the experiments the lowest recorded result was at 15 degrees angle of yaw where the result was still greater than 10 %.

Figure 6 indicates the Reynolds number dependence of the drag coefficient for Configuration 1 at zero angle of yaw; the dependence of Configurations 2 and 3 is much less significant. The drag reduction increases with Reynolds number from nearly 48% for a Re of 3.64 x10^5 to approximately 57% for a Re of 6.4x10^5.

The uncertainty of C_D was calculated to be ±1% based on the values obtained for the base configuration (Configuration 1) at zero angle of yaw.

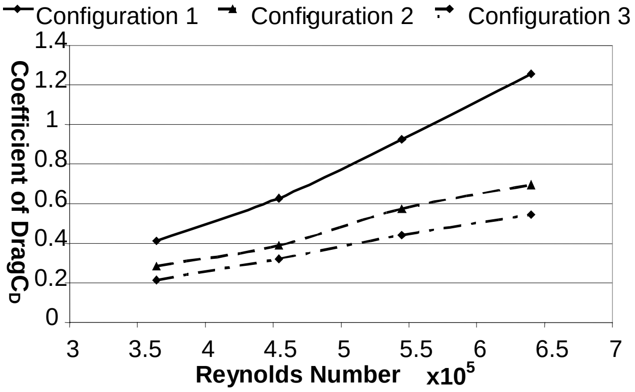


Figure 6. Reynolds Number Dependence of Drag Coefficient.

Conclusions

The present study has demonstrated the viability of using a rotating ventilator as a form of drag reducer for bluff bodies. As far as the authors are aware this is a novel application of the device which has only been used in natural ventilation.

The experimental results presented in this paper indicate a clear improvement in the drag coefficient at all angles of yaw due to the rotating ventilator. A significant reduction in drag of over 50% at zero degrees angle of yaw was achieved. This is a very encouraging finding since the relative angle of yaw of the free stream is generally small and close to zero for most transport applications however other devices may need to be attached for flows dominated by high angles. Another useful finding is that the incorporation of the rotating ventilator makes the drag of the vehicle less Reynolds number dependent. This suggests that the positive attributes of this device are applicable over the whole range of the flow speeds and thus in the case of transport vehicles will be more economic for long-haul transportation on the open road.

It should be pointed out that only one ventilator was used in this study to demonstrate the concept. However, in normal operation it is envisaged that a series of ventilators would be attached to the body. Further work is progressing.

Finally, the drag reducer considered in this study has considerable operational and economic advantages. The price of the ventilator used is quite low, approximately US$35 retail per unit, which is a very small percentage of the total vehicle cost. The device is easy to implement and costs nothing to operate. The method proposed here, therefore offers a cost effective, practical solution to a common problem of bluff body flows such as truck operation. The improved fuel efficiency resulting from the use of this concept will have significant positive impact on the bottom-line operating cost of these vehicles and also benefit the environment from reduced fuel emissions. In the case of other bluff bodies the reduced loading will result in reduced structural demands.

Acknowledgments

The authors would like to thank Edmonds Products Pty Ltd and in particular Mr Allan Ramsay and Mr Derek Nunn for their enthusiastic support in the conduct of this experiment.

References

[1] Abu-Hijeh, B.A.K., Modifying a Reattaching Shear layer using a Rotating Cylinder, *Comp. and Fluids*, **29**, 2000, 261-273.

[2] Ahmed, N. A. & Archer, R. D., Testing of a Highly Loaded Horizontal Axis Wind Turbines Designed for Optimum Performance, *Int. J. of Renewable Energy*, **25**, 2002, 613-618.

[3] Editor, Active Flow Control-Bright Prospects and Basic Challenges, *J. of Aircraft*, **38**, 2001, 401.

[4] ESDU, Fluid Forces, Pressures and Moments on Rectangular Blocks, *ESDU Data Sheet 71016,* 01 Nov 1978.

[5] Gotz, H., Bus Design Features and Their Aerodynamic Effects, *Int. J. of Vehicle Design*, Technological Advances in Vehicle Design Series, SP3, Impact of Aerodynamics on Vehicle Design, 1983, 229 – 255.

[6] Grosche, F.-R. & Meier, G.E.A., Research at DLR Göttingen on Bluff Body Aerodynamics, Drag Reduction by Wake Ventilation and Active Flow Control, *J. of Wind Eng. and Ind. Aerodyn.*, **89**, 2001, 1201-1218.

[7] Kubo, Y., et al, Control of Flow Separation from Leading Edge of a Shallow Rectangular Cylinder through Momentum Injection, *J. of Wind Eng. and Ind. Aerodyn.*, **83**, 1999, 503-514.

[8] Lajos, T., Drag Reduction by the production of a Separation Bubble on the Front of a Bluff Body, *J. of Wind Eng. and Ind. Aerodyn.*, **22**, 1986, 331-338.

[9] Mittal, S., Control of Flow Past Bluff Bodies using Rotating Control Cylinders, *J. of Fluid. Struct.*, **15**, 2001, 291-326.

[10] Modi, V.J., Moving Surface Boundary Layer Control: A Review, *J. of Fluid. Struct.*, **11**, 1997, 627-637.

[11] Modi, V.J. & Deshpande, V.S., Fluid Dynamics of a Cubic Structure as Affected by Momentum Injection and Height, *J. of Wind Eng. and Ind. Aerodyn.*, **89**, 2001, 445-470.

[12] Modi, V.J., St. Hill, S. & Yokomizo, T., Drag Reduction of Trucks through Boundary Layer Control, *J. of Wind Eng. and Ind. Aerodyn.*, **54/55**, 1995, 583-594.

[13] Munshi, S.R., Modi, V.J. &Yokomizo, T., Fluid Dynamics of Flat Plates and Rectangular Prisms in the Presence of Moving Surface boundary Layer Control, *J. of Wind Eng. and Ind. Aerodyn.*, **79**, 1999, 37-60.

[14] Rashid, D. H. & Ahmed, N. A., Study Of Aerodynamic Forces on a Rotating Wind Driven Ventilator. *Int. J. of Wind Eng.*, **27**, 2003, 63-72.

[15] Stollery, J. & Garry, K. P., Techniques for Reducing Commercial Vehicle Aerodynamic Drag., *Int. J. of Vehicle Design*, Technological Advances in Vehicle Design Series, SP3, Impact of Aerodynamics on Vehicle Design, 1983, 210 – 228.