An Investigation into Unsteady Base Bleed for Drag Reduction in Bluff Two-Box SUV's

Andrew D Lamond; Johnathan J Kennedy; Dr. Matthew Stickland
Department of Mechanical Engineering
University of Strathclyde, UK

Abstract

This paper discusses a preliminary investigation into the use of base bleed on a production SUV using CFD analysis. The paper shows the methods used in creating the computational model and conducting the analysis, and present the findings to date. The paper shows that the reduction in drag increases as the mass flow rate of air is increased when the flow is deflected at the outlet. By controlling the turbulent wake to the rear of the vehicle, it is shown in the paper that mass flow rates of under 2kg/s can reduce drag coefficient by 8.2% with an outlet on the side of the vehicle, and that a mass flow rate of under 1.5kg/s can reduce the drag coefficient by 10.7% for an outlet on the upper section of the rear of the vehicle.

The paper also discusses the feasibility of base bleed being applied to a production vehicle.

1 Introduction

As a result of pressure to increase fuel efficiency and decrease emissions, major manufacturers are researching and utilising novel methods of improving vehicle efficiency. The introduction of electric and hybrid cars has become commonplace, although the technology is still in its infancy. More traditional methods such as improving the engine design and vehicle aerodynamics could still prove to be equally effective.

It must be noted, however, that the aim of increasing fuel efficiency must be met while maintaining the practicality of the vehicle. Large 4x4 cars or sports utility vehicles, (SUV's) are desired by the consumer for aesthetic reasons, but also for their off-road ability and their high storage capacity. For these reasons, fuel economy must be increased without compromise to the power delivery from the engine or their interior space. Manufacturers of SUVs are therefore investigating innovative drag reduction methods.

1.1 Base Drag and Lift Induced Drag

Pressure drag due to flow separation constitutes more than 90% of the total drag of bluff bodies [1]. A considerable proportion of the drag of a 4x4, (that is not so prevalent in saloon cars, for example) is base drag. Base drag, a form of pressure drag, is caused as a low pressure region develops to the rear of the vehicle (the base region). This low pressure is caused by turbulence as the flow separates at the rear edges of the upper, lower and side surfaces of the car.

This is a particular problem in 4x4 vehicles due to the large flat rear section, causing a large turbulent wake. SUV's also have a tendency to have the four sides parallel to the flow at close to right angles with the rear surface. This sharp angle significantly affects the flow separation and the characteristics of the flow in the base region.

Another form of drag to be overcome is lift induced drag. It has been suggested that in the case of road vehicles, it is not possible to define an induced drag and therefore the term 'vortex drag' is preferred [2]. However, the control of these vortices will have the effect of reducing drag.

In vehicle aerodynamics, it is much more difficult to differentiate between base drag and vortex drag than it is in aeronautics and therefore the distinction is useful mainly to identify the primary source of the drag rather than to obtain quantitative predictions of each [3].

1.2 Base Bleed

Base bleed can be defined as the reduction of drag via the introduction of air or some other gas to the rearward low pressure region. Drag reduction can be achieved through repressurisation of the base region, or the air or gas may be used to control the turbulent wake of the vehicle or the trailing vortices that cause drag. Although this may still result in repressurisation within the base region, repressurisation may not be the primary influence on the change in drag.

The air or gas used will be known as 'bleed air' for the duration of this report. This bleed air can be obtained in a number of different ways: In the case of artillery shells, base bleed is often implemented through use of a gas generator to the rear of the object, an example being the M864 [4]. The most detailed study of base bleed in vehicle design to date was carried out by Howell et. al. [5]. The experimental work carried out by Howell used a supplementary air source to simulate base bleed. Following Howell's work this investigation's results were obtained by introducing a 'virtual' secondary flow to the system. However, consideration was given to bleed air being obtained through ducting from a high pressure region on the vehicle as this may be a more practical approach to a real system. This is commonly known as ventilation and is sometimes considered to be different from base bleed.

The point at which air is expelled into the base region will be known as the 'bleed outlet', positioned on the rear surface of the vehicle.

Base bleed has been studied in many different circumstances as it occurs accidentally in many practical applications. Some examples of this, sited by Wei and Chang [6], are flow between turbine blades and aerodynamic forces of high-rise buildings. It was found by Suryanarayana et. al. [7] that the drag of a sphere can be reduced significantly by passive ventilation and that simultaneously the flow field around the base is stabilised and made symmetric, leading to a reduction in unsteady aerodynamic forces.

Wood [8] and Bearman [9] have studied the effect of bleeding a secondary stream of air into the wake of a blunt based airfoil. It was reported that base bleed causes the base pressure to increase up to a certain limit and then the drag begins to increase with mass flow rate. Bearman also found that the effect is greater when the area of the jet is large when compared with that of the base. This was investigated by varying the size of bleed outlet.

Howell [5] used only a simple bluff body shape, did not simulate the effects of the wheels or the ground and was carried out entirely in the wind tunnel. This investigation aimed to expand on this work by examining vehicle base bleed using CFD on a production vehicle in motion and the effects it has on the wake of the vehicle.

2 Model Generation and Pre-processing

In order to create the computer model for use in CFD, it was decided that use of a laser scanner and 1:6 scale model should be used. Not only did this produce a representative model, but it also allowed manipulation of the surface detail and shapes of the individual surfaces. This meant that the detail could be set to an appropriate level, keeping any necessary geometry while reducing the complexity in meshing.

The CFD modelling process was carried out using the commercial CFD code FLUENT 6.3.26 and therefore GAMBIT 2.4.6 was deemed the most appropriate pre-processor and meshing tool.

2.1 Laser Scanning

Scanning was done using a Konica VI-9i non-contact 3-d digitizer, shown in figure 2.1. This is a free standing scanner suitable for scanning large objects at variable distance, accurate to 50µm. It is ideally suited to reverse engineering applications such as this.

2.2 Manual Surface creation

Using Geomagic studio 10, a manual patch layout was applied in order to create a uniform surface design. This allowed the model to be meshed quickly and easily in Gambit and was important in generating the bleed outlet designs.

The faces were carefully aligned with main vehicle features, with 422 patches used to allow sufficient retention of detail while making it easy to locate errors when meshing. The patch layout and scanned image can be seen in figure 2.1.

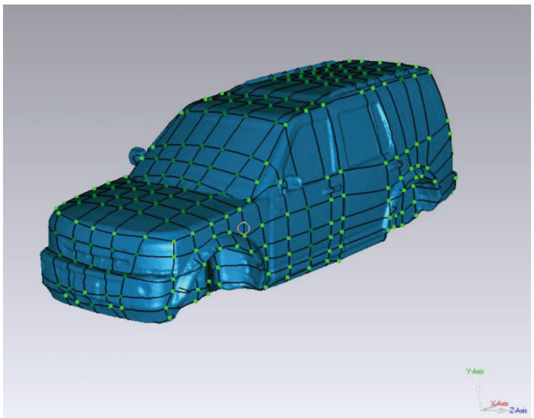

Figure 2.1 – Scanned Land Rover

The wheels were scanned using a similar process to that described previously, retaining the tread and spoke detail.

2.3 Pre-processing

The model and wheels were imported separately into Gambit 2.4.6 and aligned with a control volume. A half model was used to allow quicker solution of the model with a more refined mesh. General CFD practice dictates that in vehicle analysis, a mesh of around 2-4 million elements is considered to be a course mesh. A moderate mesh should consist of between 4 and 10 million elements, and a fine mesh will contain more than 10 million elements [10]. The mesh was graded, with a fine mesh of 3mm being applied around the vehicle, becoming gradually coarser towards the edge of the control volume. The control volume size was set according to Fluent's best practice guide for vehicle analysis [10]. The domain extended around three times the vehicle length to the front and five times to the rear. The width and height of the control volume were set so that the cross section of the vehicle did not exceed 1.5% of the domain area. With a graded mesh, approximately 2.2 million elements were used. Figures 2.2 and 2.3 show the mesh used on the car surface and the control volume respectively.

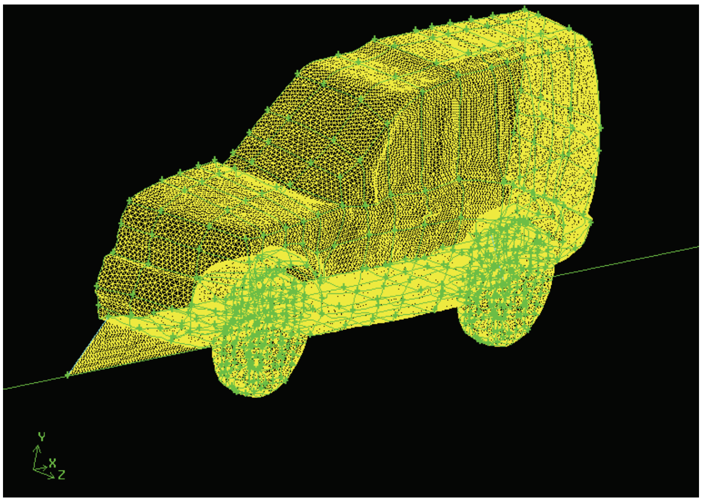

Figure 2.2 Land Rover mesh

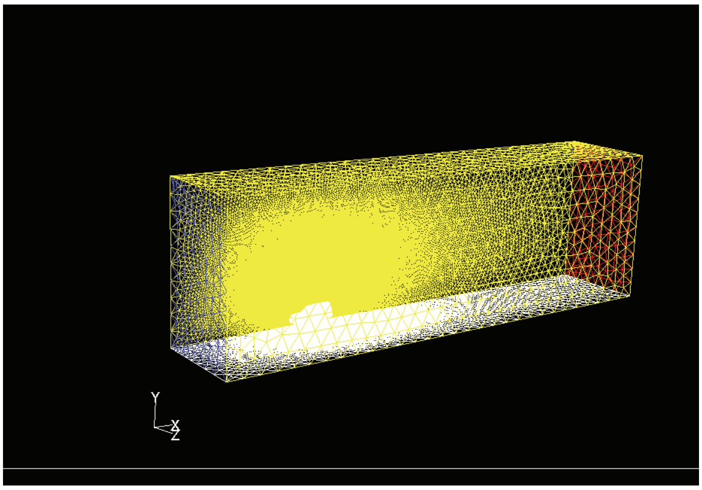

Figure 2.3 Control volume mesh

3 Results and Discussions

3.1 Initial Testing

The model was imported to Fluent 6.3.26. The model was run in Fluent for initial results. The ‘scale’ feature was used to increase the model size to allow analysis of a full scale vehicle in order to ease comparison with future studies. The convergence criteria of the residuals were set to $1e^{-5}$ and a k- ε turbulence model was used. Initial values of K and ε were calculated as follows [11]:

$ki = 4.5 \times 10^{-3} * (UIN)^2$ where UIN is the velocity at the inlet
$\varepsilon\, i = (k_i^{1.5})*0.1643/(0.09*HEIGHT)$ where HEIGHT is the flow inlet dimension.

$k_i = 2.8125$
$\varepsilon_i = 3.4442$

The inlet velocity was set at 25m/s, a speed felt comparable to motorway driving speed in the UK and therefore of particular interest. The wheels rotated at 67rad/s with radius of 0.372m. Grid adaption was used to ensure $50<y^+<500$. This was particularly necessary around the wheels and the front of the car. The model was initially solved with no base bleed.

The model was set up to allow analysis of the change to forces on different areas of the vehicle. With no base bleed applied, the half model had a total drag force at this speed of 230.65N and therefore a drag coefficient, C_d, of 0.40969 with a reference area of 1.47066m^2. There was a large pressure force (80.16N) in the positive x-direction acting on the rear of the vehicle. This draws the car rearwards and contributes greatly to the overall drag force acting on the car. To reduce this pressure value as far as possible is desired, but in doing this consideration must be given to the overall effect and the pressure forces acting on other areas of the car.

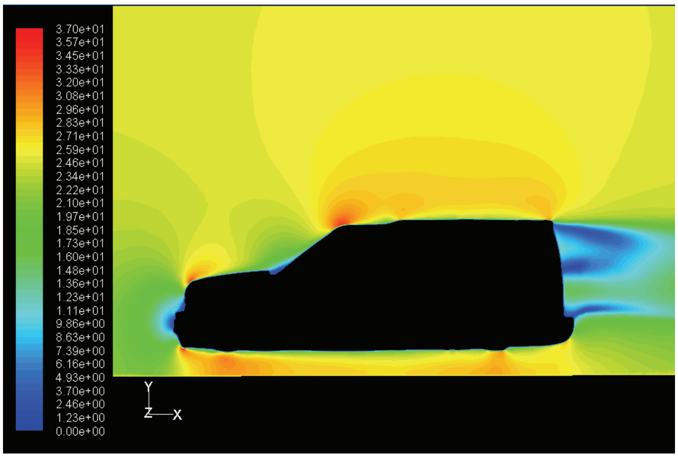

Figure 3.1 Velocity plot of airflow over standard model

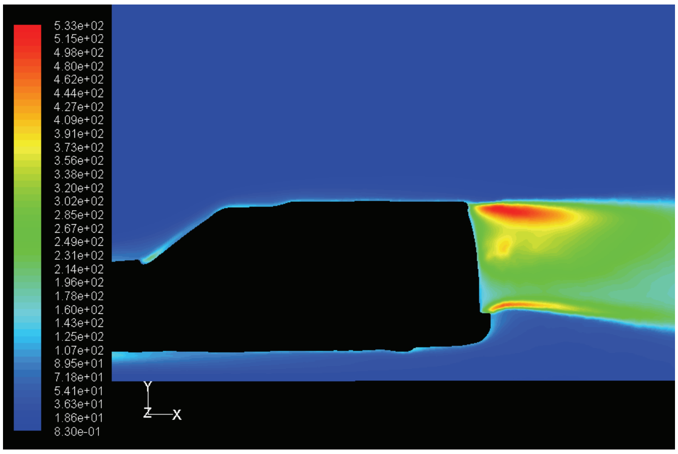

Figure 3.1 Turbulent intensity plot of standard model

Figures 3.1 and 3.2 show a large turbulent region at the upper portion of the rear of the vehicle and so investigation was carried out to see if implementing base bleed in this area could control the airflow and reduce the size of this turbulent wake, resulting in a pressure increase and drag decrease.

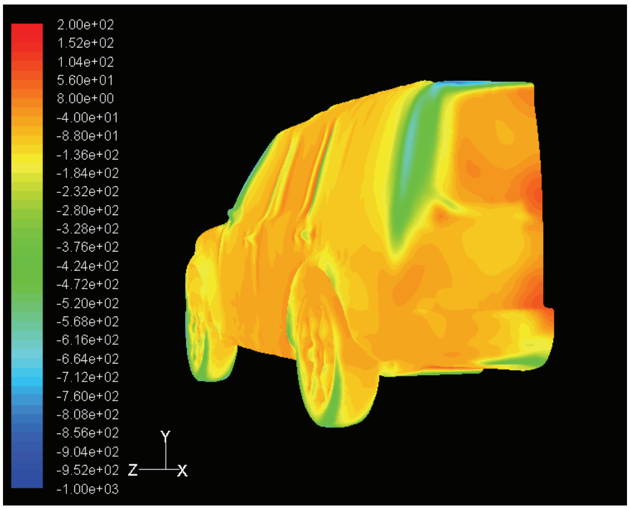

Figure 3.2 Plot of static pressure on rear of standard model

Figure 3.3 shows that there are large areas of low pressure acting on the rear of the vehicle. Areas of particular interest are the upper and side trailing edges where boundary layer separation occurs. This low pressure retards the movement of the vehicle and so is undesirable.

3.2 Base Bleed Development

Three 'bleed configurations' have been considered. J Howell [5] found that the most effective locations for a bleed outlet are at the top, side and bottom of the base region. They found that very little drag reduction was obtained using a centrally located outlet. As the bottom of the vehicle has a large bumper to which the flow remains attached particularly well (see figure 3.1), investigation was carried out with the three configurations shown in figures 3.4 to 3.6:

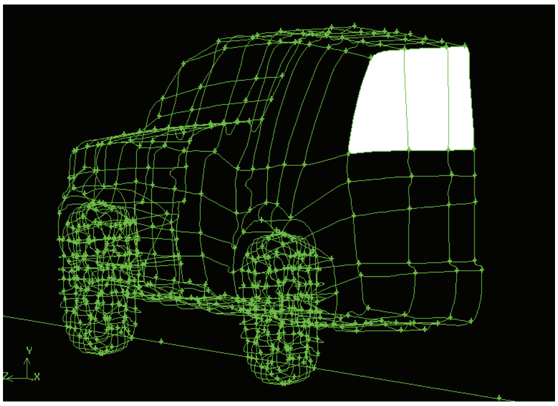

Figure 3.3 Bleed configuration 1 – outlet area 0.279m²

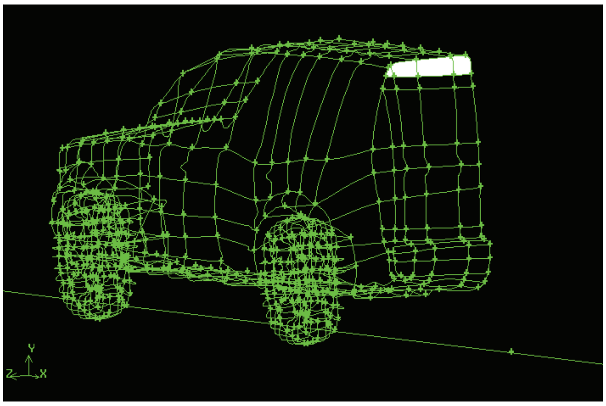

Figure 3.4 Bleed configuration 2 – outlet area 0.04733m²

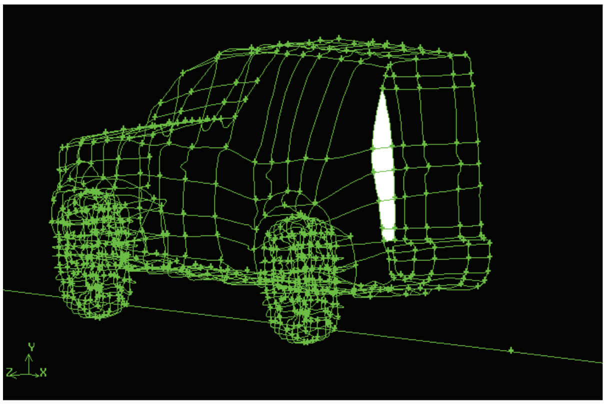

Figure 3.5 Bleed configuration 3 – outlet area 0.06388m²

The white areas in the images show the cavity through which the bleed air was blown. It should be noted that, as a half model was used, all values for bleed area and hence mass flow would need to be doubled for a full vehicle. The bleed outlets were set-up as 'velocity inlets' in Fluent, simulating a supplementary source of air. The disadvantage of this method is that the cavity that would be present in a real system was not simulated and may have a bearing on the overall result. Furthermore, the flow from the outlet does not account for the boundary layer that would be established within the duct and was therefore completely uniform.

These three configurations were investigated under varying outlet velocities (mass flow rates) as well as varying outlet flow angles to asses the effect on the flow characteristics in the base region. The outlet velocities in each case were varied between 0m/s and 25m/s. For cases 1 and 2, the flow was expelled at 90°, 60° and 45° to the Y-Z plane with a downward bias. For case 3, the flow was tested at 90°, 60° and 45° to the Y-Z plane with a bias towards the centreline of the vehicle. This is shown in figures 3.7 and 3.8 respectively.

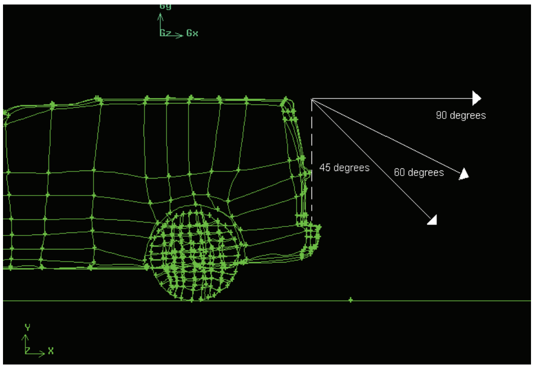

Figure 3.6 Flow angle diagram for Config. 1 and 2

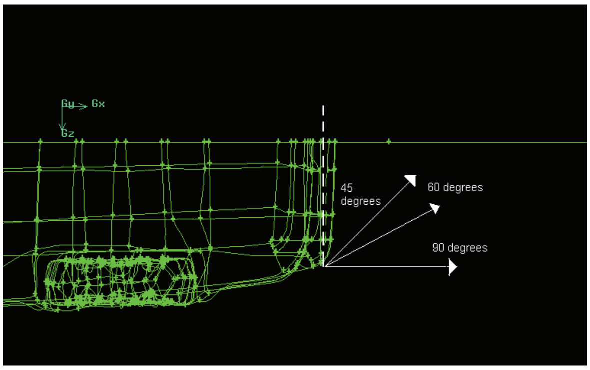

Figure 3.7 Flow angle diagram for Config. 3

3.2.1 Bleed Configuration 1

Howell [5] found that drag reduced linearly with mass flow rate and also increased with bleed area. Bleed configuration 1 was used as a test case to establish if base bleed could reduce the drag of a road vehicle in the most extreme of circumstances i.e. with a large bleed outlet area that would be unachievable in a practical sense due to the high mass flow rates required.

It was found that for a bleed velocity of 5m/s, the retarding pressure force on the rear of the vehicle decreased from 80.16N to 37.91N. Although the pressure increased over other parts of the vehicle, there was a net decrease in drag (C_d reduced to 0.393) showing that base bleed could be successfully implemented. However, this was found to be the optimum bleed velocity. As the velocity was increased, it was found that the drag started to rise (figure 3.9). This is compatible with the findings of Wood [8] and Bearman [9] and was likely due to the high mass flow rates reducing the base pressure. It was considered at this stage that the force on the rear surface dropped partly due to the decreased surface area on which the pressure acted.

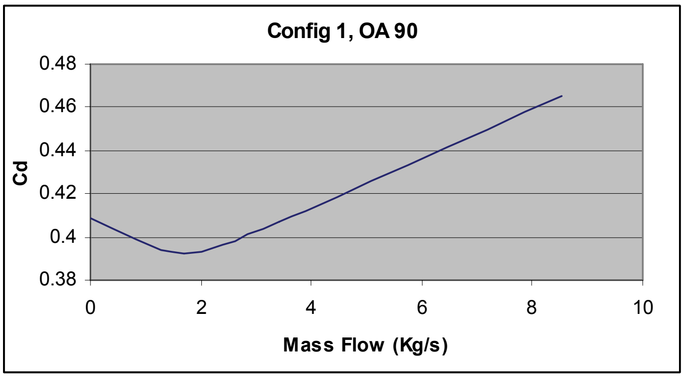

Figure 3.8 Drag values for config. 1 with flow angle of 90°

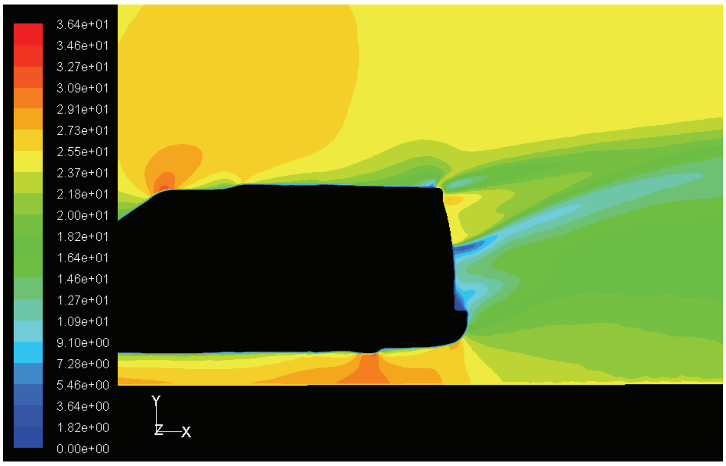

Figure 3.9 Velocity plot of bleed at 25m/s for config. 1 with flow angle of 90°

Figure 3.10 shows that base bleed in this configuration appears to draw the wake upwards, moving forward the boundary layer separation on the upper surface and increasing the overall size of the turbulent wake. This effect increases as the bleed velocity increases. As a real-life system would likely be based on passive ducting from a high pressure region and mass flow would increase with increasing forward speed, mass flow would need to be regulated as, without flow control, drag could increase as vehicle speed increased. Alternatively, the angle of the flow could be altered for a more desirable effect.

As the flow angle was altered through 45° and 60°, the results shown in figures 3.11 and 3.12 were obtained:

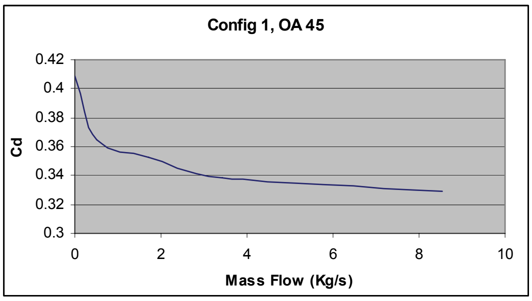

Figure 3.10 Drag values for config. 1 with flow angle of 45°

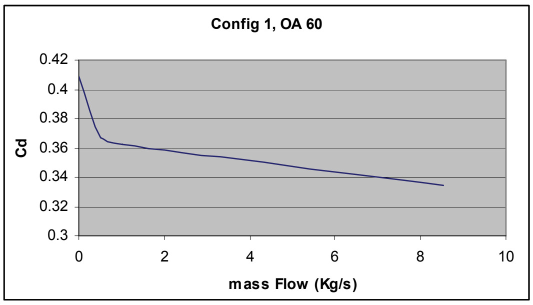

Figure 3.11 Drag values for config. 1 with flow angle of 60°

By angling the flow, base bleed was shown to significantly reduce drag. The best result achieved was a C_d of 0.329, a decrease of 19.7%. With a mass flow rate of only 0.5kg/s, the C_d was reduced to 0.365, an 11% reduction from the original vehicle. A downward outlet angle of 45° was most effective, as shown by figures 3.11 and 3.12.

3.2.2 Bleed Configuration 2

By reducing the size of the bleed outlet, the possibility of implementing such a system increases. As it has been found that significant results can be achieved from a relatively low mass flow rate, the bleed outlet area was reduced to 0.04733m^2 and low mass flow rates were studied. Similar results were found when no deflection was applied to the bleed air, with the forces ultimately increasing with mass flow. Figure 3.13 shows the results obtained with a deflection of 45° (the most successful tested):

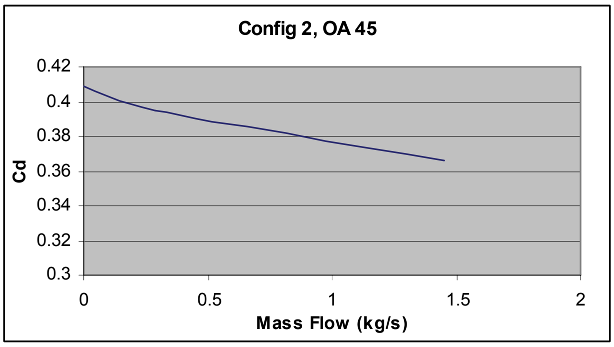

Figure 3.12 Drag values for config. 2 with flow angle of 45°

Varying the outlet velocity from 0m/s to 25m/s, it was found that C_d could be reduced to 0.366 at 25m/s with a 45° outlet angle. For this, a mass flow rate of 1.449kg/s was required. This was a decrease of 10.7% in the drag coefficient.

It should be noted that the mass flow rate must be tripled in order to obtain similar results to the larger bleed area. Therefore, for a real system to be designed, a large bleed outlet is desirable. This is in line with the findings of Bearman [9]. It must also be noted that an outlet velocity of 25m/s with a vehicle speed of 25m/s would require a inlet larger than the outlet were a ventilation system to be used as losses would be obtained in the duct and at the inlet.

These two impracticalities may mean that obtaining such high reductions in C_d would be difficult on a production vehicle, however even with an outlet velocity of 5m/s (a mass flow of 0.29kg/s) the C_d value was decreased to 0.395.

3.2.3 Bleed Configuration 3

A final configuration was investigated. Were a ventilation system to be implemented, it may be most practical to bleed air from a scoop on the underside or side of the vehicle. Therefore, a bleed outlet on the upper portion of the rear may not be as practical as one implemented on the side. There is a key difference in this configuration. Where the bleed is implemented at the side of the vehicle there are two outlets blowing towards each other and interacting.

A similar process was followed to the previous cases and it was found that, where no deflection was applied, the C_d increased to 0.46 with an outlet velocity of 25m/s. Figures 3.3, 3.14 and 3.15 allow comparison of the static pressure distribution on the rear of the vehicle under varying bleed angles.

At the sharp angle of 45°, the flow at the rear of the vehicle is accelerated around the rear edge, creating a low pressure zone towards the outside of the vehicle (Figure 3.14). However, a region of high pressure is created around the centreline of the vehicle, and overall the base pressure did increase, with a resulting C_d of 0.396 when the outlet velocity was 25m/s.

For a bleed angle of 60°, the region of fast moving, low pressure air was controlled and the sub atmospheric pressure acting on the rear of the vehicle was increased more evenly giving better results on the overall drag. This emphasises that the effect on the base region is not uniform and the importance of analysis on a real vehicle shape and not a generic bluff body.

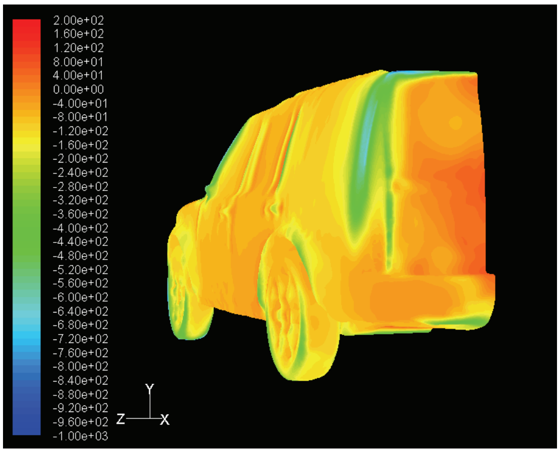

Figure 3.13 Pressure contour plot for config. 3 at 25m/s with 45° deflection

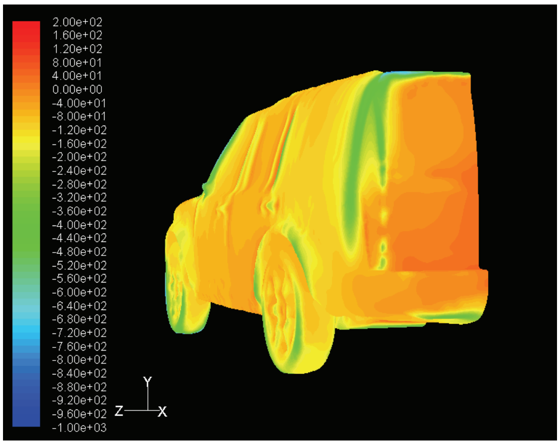

Figure 3.14 Pressure contour plot for config. 3 at 25m/s with 60° deflection

Interestingly, with a bleed mass flow of 1.956kg/s, a drag coefficient of 0.3766 was achieved, and with a mass flow of 0.235kg/s, the effect on the flow was similar with a C_d reduction to 0.3783. For this configuration, the interaction of the two bleed outlets meant that higher mass flow rates caused turbulence and ultimately did not decrease the drag coefficient linearly. This is shown in figure 3.16:

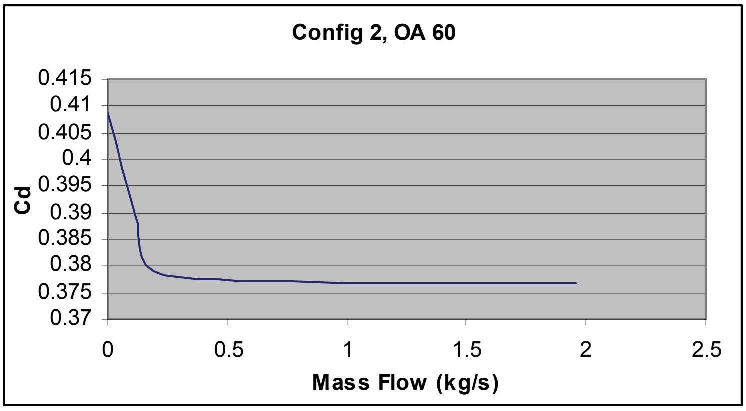

Figure 3.15 Drag values for config. 3 with flow angle of 60°

4 Conclusions

It has been shown that base bleed can significantly decrease drag when applied to the geometry of a real SUV, and that mass flow rates of under 2kg/s can reduce drag coefficient by 8.2% with an outlet on the side of the vehicle, and that a mass flow rate of under 1.5kg/s can reduce the drag coefficient by 10.7% for an outlet on the upper section of the rear of the vehicle.

It was concluded that a large outlet with a lower bleed velocity was more effective for the same mass flow rate than smaller outlet with greater bleed velocity. This is particularly relevant if applying a ventilation system where the mass flow is limited.

For the geometry studied, a bleed outlet applied to the upper portion is most effective in reducing drag. It may be difficult to implement this, however, as bleeding air from a high pressure region towards the top would be difficult. For this configuration, investigation may have to be carried out into bleeding air from an inlet on the roof of the vehicle.

It was found that bleed through the sides of the vehicle is also effective. This would allow venting from the sides, underside or roof.

It was also found that deflecting the flow has a large impact on the drag reduction achieved, and that the flow angle should be optimised for any system implemented. As the flow around the rear of any real vehicle is largely non-uniform, investigation into the optimum angle would have to be carried out for the vehicle in question.

Were this system to be implemented, the cavity of the bleed outlet may in fact increase drag reduction further [5]. Wind tunnel testing or a more detailed computer model would be required to investigate this.

5 References

1. Experiments on the flow past spheres at very high Reynolds numbers; Aachenbach E.; 1972; Fluid Mech. 62, 209-221

2. The effect of base slant on the flow pattern and drag of three-dimensional bodies with blunt ends; R. T. Jones; in Aerodynamic drag mechanism of blunt bodies and vehicles, Plenum Press, NY, 1978

3. Motor vehicle dynamics: modeling and simulation; Giancarlo Genta; Published by World Scientific, 1997; ISBN 9810229119, 9789810229115

4. Predicted Flight Performance of Base-Bleed Projectiles; James E. Danberg and Charles J. Nietubicz; U.S. Army Ballistic Research Laboratory, Aberdeen Proving Ground, Maryland

5. Aerodynamic Drag Reduction for a Simple Bluff Body Using Base Bleed; Jeff Howell and Andrew Sheppard -Land Rover; Alex Blakemore -Loughborough University; SAE Technical Paper Series 2003-01-0995

6. Wake and base-bleed flow downstream of bluff bodies with different geometry; Chin-Yi Wei, Jeng-Ren Chang; 2002; Experimental Thermal and Fluid Science 26; 39–52

7. Bluff-body drag reduction by passive ventilation; G. K. Suryanarayana, Hemming Pauer, G. E. A. Meier; 1993; Experiments in Fluids 16, 73-81

8. The effect of base bleed on periodic wake; Wood, C. J. 1964; Royal Aeronautical Society, Vol. 68, 477-482

9. The effect of base bleed on the flow behind a two-dimensional model with a blunt trailing edge; Bearman, P. W; 1967; The Aeronautical Quarterly, 207-224

10. '3. Meshing', *Best practice guidelines for handling Automotive External Aerodynamics with FLUENT*, Version 1.2; Lanfrit,M; 2005; Fluent Deutschland GmbH Birkenweg 14a64295 Darmstadt/Germany,19,

11. TURBULENT FLOW - 16597 Computational Fluid Dynamics notes; T Scanlon; University of Strathclyde