Research into the Installation and Optimisation of Turning Vanes and Drag Reduction Technology applied to the Land Rover Discovery MK3 Vehicle

Johnathan James Kennedy, Andrew Lamond, Dr Matthew Stickland
Faculty of Mechanical Engineering
University of Strathclyde, UK

Abstract
Previous industry based research has been carried out on the drag reduction of the Land Rover discovery MK3 based on CFD models. It has been noted that the cumulative drag coefficient for the open cooling case is lower than that for the closed cooling case, until the cooling flow re-emerges into the bulk flow. This indicates that the energy loss incurred from passing through the vehicle's heat exchangers is less than that of the loss incurred by the fluid negotiating a path around the large bluff front section. This discovery led to the idea of using front turning vanes to control flow. It has been found during the course of this research that turning vanes can significantly decrease drag when placed on specific areas that display an adverse pressure gradient on the surface of the vehicle, by as much as 13.5%. Rear/roof turning vanes have been found to exhibit the greatest reduction in $C_D$ compared to turning vanes placed towards the front of the vehicle.

1. INTRODUCTION

Auto makers have had a keen interest in vehicle aerodynamics for over 70 years; since the introduction of the Chrysler ‘Airflow’ in 1934. But even in these early days Aerodynamic styling was viewed as a dark art, and rarely left the drawing board. The main exponent of aerodynamic styling and structure was the auto racing sports; it was quickly discovered that the better aerodynamically designed car could win, and win often. Even after the realisation of the major Auto companies track successes, due in no uncertain terms to streamlining leading to decreased drag, auto companies were still reluctant to incorporate these aspects in everyday consumer cars. But since these early days of vehicle aerodynamic design priority has incrementally shifted in this direction; the need to improve fuel economy has now become paramount. The easiest method of fuel economy is to create smaller light weight cars, with small fuel efficient engines – the predicament arises when the consumer is known to predominantly want 400 hp sports cars and seven passenger SUVs, while also obtaining good fuel efficiency.

1.1 Aerodynamic drag and Mileage effect
"For a full-size truck, a change in drag coefficient of 0.01 is approximately equal to an improvement in fuel economy of 0.1 mpg on the combined city/highway driving cycle," says
GM’s Schenkel. He adds: "The same drag coefficient reduction can improve a car's fuel economy by approximately 0.2 mpg."\(^1\)

Such improvements are a major boon to the average car buyer and operator, but major drag reduction on the sizeable and boxy SUV market can greatly increase the popularity and affordability of already costly, diesel hungry designs. The drag Coefficient is a function of air density and vehicle speed, as well as the ‘wetted’ frontal area of the vehicle. Drag is proportional to the square of the velocity, and power required is equal to the cube of the velocity. In virtually all cases, apart from in the infinitesimally thin plate, the flow separates from the body of the structure at some point. In general the structures that produce a boundary layer separation induce a disturbed down stream eddy flow, termed the wake. As a result of the energy dissipated by the highly turbulent motion in the wake the pressure there is reduced and the pressure drag on the body is thus increased\(^2\). From this premise a divergence in body shape and drag reaction can be asserted: Streamlined bodies and Bluff bodies are the terms used to describe a body that has a very small wake and thus a decreased pressure drag, and a body that has a large pressure drag and separation taking place over much of its surface area, respectively.

1.2 Boundary Layer Control
When considering airflow around an object such as a car, there are regions where air viscosity is important and regions where it can be neglected. In a thin layer of flow near the body surface, (the boundary layer), the effect of viscosity is important. However outside this layer, the flow can be modelled as being inviscid (i.e. zero viscosity)\(^4\). The boundary layer can be differentiated into two specific types, laminar and turbulent. In laminar flow airflow progresses in sheets (laminae, or layers), contrastingly turbulent flow represents a mixing within layers. Flows initiate in a laminar way, and then transition to turbulent flow either results naturally or is encouraged with the use of designed roughness (vortex generation). As was mentioned earlier the drag on a vehicle is greatly dependent on the identity of the boundary layer, and especially the point of separation. Air-flow that precisely follows the shape of the vehicle is referred to as ‘attached’ flow; if the flow departs from the vehicle shape it is defined as ‘separated’ flow, maintaining the ‘attached’ flow has a great influence on drag. Much can clearly be done to diminish aerodynamic profile drag; box shaped geometry can be incrementally changed so that it adheres morphologically to the most streamlined shape possible, a tear-drop. It has been shown theoretically\(^5\) that a ‘cowl’ or vane redirection can reduce the drag of a blunt body (such as an aircraft engine). The external vanes allow airflow that would otherwise shear off due to adverse pressure gradients and create a turbulent wake, to remain forcibly attached around sharp corners.

1.3 Current Turning vane application
Turning vanes, predominantly side turning vanes (fig.1) are used in 68% of all long hall and medium hall freight vehicles according to Dept. for transport data\(^10\). Usually placed below the Cab window, they cover sharp edges and help to reduce build up of road film and dirt. They have an addition cost of £100 and a payback of 0.7 years in terms of aerodynamic fuel savings, (Data recorded for the average long/medium hall freight transport vehicle).
The majority of the air passes around the exterior of the vanes and evidence suggests that any drag reduction obtained derives from the increase in the effective radius of the cab edges to a level that is sufficient to ensure that the flow does not separate\textsuperscript{10}. Preliminary CFD studies have shown that an increased radius has a similar effect to turning vane inclusion on a simple brick model within fluent\textsuperscript{(fig.4, & 5)}. Attached flow is also maintained and turned forcibly by the inside structure of the vane. Indicating that the vane has two distinct purposes: 1, providing a greater effective radius around a sharp corner; 2, delaying boundary layer separation due in part to adverse pressure gradients.

The department for transport lists the following turning vane points for consideration\textsuperscript{10}:

- If the cab edge is sharp or has a small radius, vanes can provide a slight reduction in drag
- Suitably located vanes can reduce dirt deposition on the cab side windows or doors
- Improved cab appearance with less need for washing
- Drag can be increased if the cab edges are already well-rounded

In this simple fluent flow case a half model of a brick (2x2x4m) was simulated travelling at 25m/s above a ground moving also at this velocity. The brick was tested with sharp corners, rounded corners and also corners with a front turning vane applied. The boundary layer is clearly seen (fig.2, and 3) to be rapidly separating at the sharp corner, the pressure gradient was so adverse that the boundary layer flow has been pushed back creating a zero velocity flow reversal. The faster moving air-flow needs to be re-attached with minimum turbulence to reduce the drag coefficient. In the sharp angled block study the fast moving layer never reaches the re-attachment stage (due to the block being too short) which adds to the rear drafting effect, effectively ‘sucking’ the block backwards.

In the cases pictured in fig.4 and fig. 5, the radius case and the turning vane case appear to both decrease the drag coefficient in the same manner; through increased external radius size leading to a smooth and gradual transition and increased velocity around the corner of the brick model. The 200mm radius case in fig.5 is potentially more effective than the turning vane case in fig.4, but either solution is acceptable when compared to the sharp corner of fig.3.
As seen in *table-1* the results hinge conclusively on the extended radius produced in both the front turning vane case and the 200mm radius case, this indicates that the drag decrease produced by a turning vane can adequately be produced by an increase in external radius. Implying that for adequately rounded structures on the *Discovery* there will be no benefit from turning vane addition, and potentially the addition of vanes may lead to a drag increase.

### 2. CFD MODEL GENERATION

#### 2.1 Geometry creation

After the preliminary simple block CFD runs, turning vanes were shown to produce a drag reduction, the aim of this paper is to illustrate the results of turning vane shape and placement on a scale reproduction of an SUV. A 1:6 scale production accurate model of a *Land Rover Discovery* was purchased, in order to create a CAD model. A *Konica Minolta* laser scanner was used to create a point cloud (*fig.6*) that was then read into *Catia V5*. The point cloud was converted into NURB surfaces (*fig.7*) which allowed the adaption of the surfaced structure within the CAD package. This was then viable to be transferred (as a *.model* file) to the pre-processor program, *Gambit* and then the commercial CFD code *Fluent*, and also successfully re-designed in *Catia V5*. This led to a successful concurrent engineering process with regard to the simulation aspect of this project, and the usage of leading industrial standard design and validation software.

The polygon editing tool (*Geomagic*) was used to create the point cloud shown in *fig.6* via a stitching process of the scans. Each scan was taken at ~20 ° intervals turning the car
through 360º (using the bottom of the chassis as a datum); the same process was repeated through pitch and yaw, using the same datum. This method produced a lot of images, but the repeated data on the previous scan allowed for accurate supplemental scan stitching, and eventually a highly detailed point cloud (fig.6). The aim was to refine the number of NURBS patches to allow for ease of meshing, but also retain the surface detail in order to provide CFD results that will accurately correlate with the actual vehicle morphology.

Fig. 8 shows the final Catia V5 CAD model produced. Parts of the model were simplified for ease of meshing and economical solving time; the adaptations were executed at the point cloud stage, excess windscreen wiper detail and wing mirror details were removed for the CFD model.

Fig.10, shows the CAD model obtained from the Laser scanning. This model is fully capable of adaption and redesign within the CAD programme, and allows a reasonably accurate and transferable fluent analysis model, that can be easily imported. CFD, CAD and scale model morphology can be seen comparing reasonably when comparing fig.9 with fig.10.
2.2 Meshing, simulation and experimental design

All the Computer modelling in the project was to aid the creation of a valid CFD model in Fluent. The CFD model produced for this project was created to represent the Land Rover Discovery Mk 3 disco, but also be simplified to the extent that runs were conducted smoothly and accurately and economically with respect to time in fluent.

It is common practice to create half-models for CFD runs, halving run time. It was found that creating a half model of the Land Rover Discovery was simple; using the Boolean remove function with respect to the control volume produced an accurate half model (fig.11).

The finest mesh was used following wind tunnel mesh convention, with respect to vehicle velocity (25m/s), (fig.13). The use of a large control volume (fig.14) necessitated mesh grading, large cells refining to small cells on the vehicle surface. High CD,A values were obtained in early simulations and where the result of a coarse mesh, applied to a vehicle within a control volume that was limiting surrounding airflow; the inlet and outlet boundaries of the earlier computational domain were too close to the model hence causing a non
uniform flow-field at the boundaries. The optimum simulation was refined using the following method:
TGrid tet-mesh was used to mesh the control volume after the surfaces of interest were accurately meshed using the finest mesh in the study (fig.12). The surrounding control volume was graded using the edge mesh function; growth rate was closely monitored and set at no greater than 20% to allow for a gradual size increase in the important areas.

Figure 13 - Graph estimating surface mesh reference length, Lanfrit (2005)

The graph above indicates mesh sizes with respect to flow velocities. A mesh size of 3mm was selected for the vehicle surface, to decrease vehicle angularity and increase the CFD accuracy.

This grading system was important as it proved impossible to create a uniform mesh size that would be convergent with respect to the accuracy of the results, time for analysis, and adequate size of control volume. Average run time was still approx. 14 hours per analysis, using RNG differential viscosity, moving road and accurately rotating wheels.

2.3 Car half model with RNG k-epsilon turbulence
The RNG turbulence model within fluent is a feature that can conduct a turbulent viscosity model with greater accuracy and greater reliability than the standard k-epsilon model, for a wider class of flows. The aerodynamics of the Discovery contain a variety of different surface shapes, and thus form drag differs across the surface of the car; the standard k-epsilon model only accounts for a fully turbulent situation, where as the RNG model significantly improves accuracy for strained flows (through an addition to the epsilon equation) and also turbulent swirling flows. Low-Reynolds-number effects are accounted for as well as high Reynolds effects, increasing accuracy in the attached and un-attached boundary layer regions of the vehicle form. The RNG model is viewed as a statistical refinement to the standard k-epsilon model. Improvements obtained from the RNG K-epsilon model were attributed to the better treatment of near wall turbulence effects.
According to agreed best practice for vehicle simulation, the size of the domain for the external aerodynamic analysis of a vehicle should be at least 3 vehicle lengths in front of the vehicle and 5 vehicle lengths behind. With reference to data gathered on vehicle analysis the vehicle should not be greater (in cross-sectional area) than 1 – 1.5% of the total control volume's individual surface areas. With this standard working data, the control volume (fig.14) had dimensions of: length: 8m; height: 2.5m; breadth: 1.6m. The whole area can easily be scaled to actual size in Fluent. All Fluent runs were carried out in actual vehicle 1:1 scale. Wall boundary zones were created for both the front and rear wheels and the wheels were set to rotate at an angular velocity of 67.204rad/s, which relates to the transitional mesh of the road and the immersing fluid velocity (i.e. 25m/s).

3. CFD RESULTS AND MODEL RUNS

3.1 Origin model

The initial fluent solution carried out was named the ‘datum’ and the model used was that originally scanned into the computer. This first run gave a base line for adaption and refinement via cumulative trial and error. The full vehicle vector plot shown in fig.15 illustrates in greater clarity the high and low velocity fluid areas; bluff front, the wheels and the windscreen pillar. The low velocity zones are seen predominantly behind the front wheel arch and at other points that halt or restrict air velocity. High pressure low velocity fluid pockets could be bled to decrease drag. When comparing the velocity vector image (fig.15) with the static pressure images (fig.17, and 18) the areas of low fluid velocity and high pressure can be seen (this is obviously drag producing). Areas of high fluid velocity and very low pressure (as seen on the wind screen roof junction and also the windsreen pillar area) are main indicators of adverse pressure gradients, leading to a detachment of the boundary layer, turning vanes in these areas should produce an aerodynamic bonus to the vehicle. It can also be seen that the turbulence of the front wheel is producing a wake that prevents the air flow from taking advantage of the streamlined central door section. The air-flow over the door section is low velocity and re-circulating, indicating a significant delay in boundary layer re-attachment after the front wheel turbulence.
The wheels are turning from the centre points (fig.16) the main body of the car is seen to be static, due to the airflow moving past the stationary car, as per standard wind-tunnel simulation. The velocity vector plot (fig.15) shows the fluid path lines across the surface of the vehicle.

**Table – 2 Aerodynamic force plot**

<table>
<thead>
<tr>
<th>Zone name</th>
<th>Pressure Force</th>
<th>Viscous Force</th>
<th>Total Force</th>
<th>Pressure Coefficient</th>
<th>Viscous Coefficient</th>
<th>Total Coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>wall</td>
<td>195.29844</td>
<td>13.83048</td>
<td>209.12892</td>
<td>0.5402993</td>
<td>0.03405315</td>
<td>0.57435258</td>
</tr>
<tr>
<td>wheel_front</td>
<td>28.719957</td>
<td>1.197826</td>
<td>29.917782</td>
<td>0.054117805</td>
<td>0.001315114</td>
<td>0.0554299497</td>
</tr>
<tr>
<td>wheel_rear</td>
<td>18.024881</td>
<td>0.6915413</td>
<td>18.716423</td>
<td>0.04892265</td>
<td>0.001590678</td>
<td>0.05052338</td>
</tr>
<tr>
<td>net</td>
<td>234.54786</td>
<td>15.227682</td>
<td>249.77646</td>
<td>0.6126993</td>
<td>0.03978864</td>
<td>0.65247776</td>
</tr>
</tbody>
</table>

**Table - 3**

<table>
<thead>
<tr>
<th>Projected area (A)</th>
<th>1.55 m² (half model)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drag force (F_d)</td>
<td>249.77 N (half model)</td>
</tr>
<tr>
<td>Coeff. Drag (C_d)</td>
<td>0.42</td>
</tr>
</tbody>
</table>

Total force (table-2) and C_d (table-3) were lower than first predicted in published data for the *Discovery* in a non-heat exchanger flow model, but for the matter of determining a vehicle drag reduction, it is of little importance, this model was used as the ‘datum’ base study and % increase or decrease of C_d was assessed with regard to this more accurate moving wheel model.

**4. APPLIED DRAG REDUCTION DESIGN RESULTS**

**4.1 Side turning vane**
A number of differing side turning vane designs were generated and applied to the areas indicated in fig.17, and fig.18, i.e. the low pressure high velocity zones, i.e. the zones indicating adverse pressure gradients leading to an increased chance of boundary separation. Fig.19 and fig.20 show turning vane studies on the bluff front fender and windscreen column respectively. The vane (fig.20) is inline with the pillar and provides a small channel (between the vehicle and the vane) for air to manoeuvre around the bluff corner, the external side of the vane supplies a smooth curvature, which allows the adjacent air-flow to ‘stick’ to the surface.

Table - 4

<table>
<thead>
<tr>
<th>Vane Description</th>
<th>CD</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bluff front side vane (fig.19)</td>
<td>0.43</td>
</tr>
<tr>
<td>Windscreen column vane (fig.20)</td>
<td>0.39</td>
</tr>
<tr>
<td>Windscreen roof vane (fig.23)</td>
<td>0.43</td>
</tr>
<tr>
<td>Rear step vane (fig.21)</td>
<td>0.42</td>
</tr>
<tr>
<td>Side rear vane (fig.22)</td>
<td>0.41</td>
</tr>
</tbody>
</table>

* With respect to ‘datum’ model (fig.16)

4.2 Rear roof turning vane

The rear vane (fig.24) is placed on the lowest pressure area of the vehicle (w.r.t fig.18) and one of the sharpest angles on the vehicle, nearly 90° vertical drop from the roof section of the vehicle.

The rear turning vane shows air-flow forcibly maintained towards the rear of the vehicle (fig.25) and then redirected; the turbulent separation point is not premature on the top side of the vane, although, vortices can be detected in the velocity plot in fig.25

Table-5.

| Coeff. Drag (CD) | 0.37 (12% drag decrease)* |

This is a 12% drag decrease when compared to the CD of the datum model (table-5).
The air-flow has been maintained around the sharp corner with use of the turning vane, (fig.25). The rear roof turning vane has seemed to increase the fluid velocity behind the vehicle, due to increased fluid mixing and vortex creation. The rear turning vane also can be seen to be acting as a vortex generator (VG) when compared to the datum model (fig.26); vortex generation is simulated in section 4.3.

Table - 6

<table>
<thead>
<tr>
<th>Configuration</th>
<th>$C_D$</th>
<th>Drag Percentage</th>
</tr>
</thead>
<tbody>
<tr>
<td>Large inlet/nozzle outlet</td>
<td>0.45</td>
<td>6.67% drag increase</td>
</tr>
<tr>
<td>Narrow inlet/large outlet</td>
<td>0.40</td>
<td>4.77% drag decrease</td>
</tr>
<tr>
<td>Rear cowl (fig.28)</td>
<td>0.39</td>
<td>7.15% drag decrease</td>
</tr>
</tbody>
</table>

Table 6 shows the rear vanes tested and their respective $C_D$s. Rear cowl was effective, and an inlet roughly the size of the attached boundary layer thickness was observed to provide the optimum drag reduction.

4.3 Rear vortex generation and drag reduction

A turbulent boundary layer also delays flow separation in certain circumstances. Or even causes the re-attachment of separated flow. This design (fig.29) tests vortex generation, and the drag relieving effects it is seen to produce. Fig.29 is a relatively basic vortex generating structure placed at the rear of the discovery. It was found that the optimum height of the VGs is almost equivalent to the thickness of the boundary layer (15 to 25 mm) and the optimum method of placement is to arrange them in a row in the lateral direction 100 mm upstream of the roof end at intervals of 100mm. The VGs are not highly sensitive to these parameters and their optimum value ranges are wide. Better effects are obtained from delta-wing-shaped VGs than from bump-shaped VGs. The vortex generators were placed and designed at (100mm) intervals as per the best practice investigation.
In this velocity magnitude plot (fig.31) the air speed has been increased in two distinct pockets behind the vehicle, this is probably due to increased turbulence and vortex creation induced by the vortex generators. High areas of turbulence can be seen resulting from the VGs (fig.30). Air velocity has effectively been doubled to a maximum of 16.5m/s, from the datum case (fig.31 indicated by circle) indicating energy dissipation in the form of turbulence.

Table - 7

| Coeff. Drag (C_D) | 0.40 |

The C_D indicates a 4.8% C_D decrease (table-7) from the accurate datum model (fig.26), indicating a potential area that could result in better aerodynamics with little redesign or cost.

5. CONCLUSIONS

- A drag decrease of 12% has been found while using a turning vane structure on the rear roof of the vehicle (fig.24).
- The turning vane effect does not decrease drag for already well rounded vehicle components (e.g. the air-flow remains attached to the under tray in fig.25 and 26 until the sharp corner), (also side turning vanes are ineffective on the Discovery due to the suitable radius currently employed (fig.4, 5 & fig.19, table-4)).
- Evidence suggests that any drag reduction obtained derives from the increase in the effective external radius of the turning vane (fig.4, 5).
- Inappropriately placed turning vanes have at best no discernable effects, and at worst actually increase drag, (table-4), (table-6), (fig.27).
- Rear vortex generators when placed as designed (fig.29) have a drag reducing ability through prematurely forcing transition to the turbulent boundary layer (fig.28), and reducing rear pressure drop. Providing a 4.8% C_D decrease from datum model of the Discovery (table-7). This turbulent layer is faster moving across the body of the vehicle and accounts for the observed viscous force drop.
- Optimum size of rear turning vane depends on the angle of the structure and the related adverse pressure gradient, the smoothness of the vane external radius allows adjacent airflow to ‘stick’ to the surface, maintaining attached flow.
6. REFERENCES

8. KOIKE,M; NAGAYOSHI,T; HAMAMOTO,N. (2004), ‘Conclusions’, Research on Aerodynamic Drag Reduction by Vortex Generators, Mitsubishi Motors technical review No.16