NUMERICAL STUDY OF FLOW OVER A BLUFF BODY WITH DRAG REDUCTION DEVICES

A .Abikan, Y. Lu and Z. Yang

Department of Mechanical Engineering and the Built Environment
College of Engineering and Technology
University of Derby, Derby, UK

ABSTRACT

A numerical study of flow over a bluff body with drag devices has been carried out using the Reynolds-Averaged-Navier-Stokes (RANS) approach and the performance of three turbulence models, the realizable *k-ε,* the *SST k-ω* and a Reynolds Stress Model (RSM), has been assessed. The predictions of both the mean and turbulent quantities agree reasonably well with the experimental data and the RSM gives the best overall predictions. A qualitative comparison between the predicted flow field and the measurements in the near wake region has also been presented and a reasonably good agreement is obtained. It is demonstrated that the RANS approach is capable of producing reasonably good results for this kind of flow although it is inherently unsteady due to vortex shedding in the wake region.

INTRODUCTION

The aerodynamics performance of heavy vehicles is relatively poor compared against other ground vehicles due to their un-streamlined body shapes. Heavy trucks usually have a boxy shape with many sharp edges [1, 2], which causes massive flow separation and higher aerodynamics drag.

Aerodynamic drag reduction is one of the major concerns of heavy truck design since it is directly related to fuel consumption with approximately 4% fuel savings by a 20% aerodynamic drag reduction at an operating speed of 105 km/h for a tractor-trailer weighing 36 tons [3]. The aerodynamic drag distribution for a truck is usually split as: the front face of a tractor generates 25% of the total drag, the gap between the tractor and trailer generates 20% of the total drag with the rear of the trailer contributing another 25% of the total drag, and the rest 30% of the total drag is due to the underside of the truck [4].

Lots of studies have been carried out experimentally and numerically with the aim of reducing drag of heavy trucks. A model of a simple ground vehicle called the 'Ahmed body' which is a generally accepted reference model was developed [5]. The experimental work showed that the base slant angle of the body affected the drag results. At a base angle of zero the pressure drag result was obtained primarily as a result of the rear flat plate. An increase in the base angle resulted in decrease pressure drag from the base. This experiment was repeated [6] at realistic flow conditions and the Reynolds number effects were examined. It was observed that the shedding from the upper and lower corners is non-symmetric due to the effect of the ground. The effects of ground clearance, moving ground, splitter plates and cavities were investigated experimentally [7]. It was discovered that there was no significant difference between the moving and stationary ground plane for higher ground clearance (0.48). At a smaller ground clearance (C/H= 0.08, H is the height of the model) there was an 8% change in overall mean pressure and at an even smaller ground clearance (C/H=0.04), there was a significant change in base pressure distribution. The experiment showed that a 50% reduction in fluctuating velocities can be achieved in the near wake with the use of cavities. The effect of cavity was also investigated [8, 9] along with effects of plates and boat-tail devices. The experiments showed that configurations with cavity suppressed the pumping action of the shear flow close to the base. The configurations with the plates and the boat-tail had a significantly narrower recirculating flow region due partly to the smaller effective base area at separation and partly to the rapid growth of the shear layer in the underbody flow region. The configuration with both a boat tail and cavity produced the most drag reduction of 50%.

Majority of the numerical investigation studies on the wake region of simplified ground vehicles such as [10] [11] has focused on the unsteady nature of the wake region. The studies that are carried out on simple ground bodies using the RANS approach usually avoid the detailed flow field analysis of the near wake region and focus primarily on comparison of drag coefficient results.

This study will assess the performance of three turbulence models – realizable *k-ε, SST k-ω* and a RSM for a simplified ground vehicle case with drag devices and also perform flow field analysis in the near wake region.

GEOMETRY AND NUMERICAL DETAILS

The computational domain matches the dimensions of the wind tunnel [8], with an upstream length of 390mm and a downstream length of 1420mm. As the averaged flow field is symmetrical, the domain width is 305mm. This is half the width used in the experiment with a symmetry boundary condition used. The ground clearance used is 20mm which when scaled up, is quite similar to those used in trucks. The Reynolds number (Re=UH/ν) was $7.6x10^5$ based on the model length.

Figures 1-4 depict the base model used along with the three drag reduction devices used in [8]. The height (H) of the model was 100mm, and the width (W) was 140mm. The drag reduction devices are extended up to 50mm, with the boat-tail configurations drafted out at an angle of 11.30°.

Figure 1 Base model

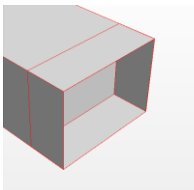

Figure 2 Case 1

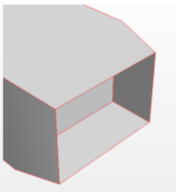

Figure 3 Case 2

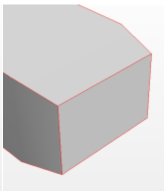

Figure 4 Case 3

Boundary Conditions

A symmetry boundary condition was used to simulate half of the wind tunnel domain. The lower, top and side boundaries were set as a no slip wall.

The wind tunnel in the experiments produced a low turbulence intensity of 0.2% and a constant velocity profile of 108km/h was specified at the inlet. A turbulent length scale of about 7% of the inlet height was specified. A constant static pressure was specified at the wind tunnel outlet.

Mesh generation

A prism layer volume mesh was used on the vehicle model and on the ground as the gradients of the physical quantities here are of interest to this study. The prismatic growth ratio used was 1.2 with the first cell height specified as $1.1x10^{-2}$mm. The first cell centre was located at $y^+ \leq 1$ to avoid using a wall function and hence capture the flow field accurately in the near wall region. The CFD code used was STAR CCM+.

RESULTS AND ANALYSIS

Mesh Independence

In all CFD simulations, grid independence tests are important to minimise the numerical errors without wasting computational resources. Figure 5 presents the normalised axial profiles on the symmetry plane at a streamwise location of 3H from the back plate of the base model with three grids (250K, 800K and 1M cells). It can be seen that there are some discrepancies between the coarse grid (250k cells) results and the results by the medium/fine mesh in the near wall region, especially the peak values. The results obtained using the medium grid (800K cells) and the fine grid (1M cells) are almost identical and hence there is no need to refine the grid any further with the fine grid being used for the current study.

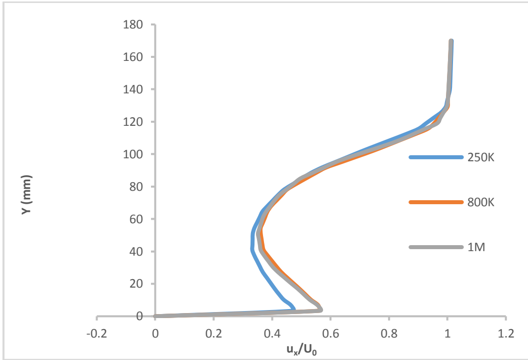

Figure 5 Velocity profile at a distance of 3H from the back plate of the base model.

Turbulence Model Comparison

Figure 6 shows the velocity profile in the wake at a distance 1.5H downstream of the back plate of the model on the symmetry plane. It can be seen that a reasonably good agreement has been obtained between the predictions by all three turbulence models and the experimental data apart from the near wall peak value, which is most accurately captured by the RSM. Both the realizable *k-ε* and the *SST k-ω* models over-predict the peak value. Nevertheless all three turbulence models over-predict the free stream velocity value and this is due to a stronger reverse flow region being predicted. Overall the RSM performs best and hence has been used for the rest

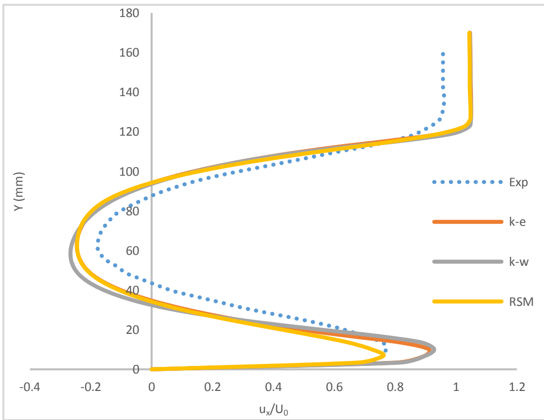

Figure 6 Comparison of velocity profiles by different turbulence models for case 1

of the current study.

Drag Coefficient

The predicted and experimental drag coefficients (C_d) are given in Table 1 below. The predicted results show a 4%

drag reduction for case 1 compared against a 12% reduction from the experiment. For case 2 a 41% drag reduction is predicted while the experimental data show a 48% reduction, and for case 3 the predicted drag reduction is 38% which is very close the measurement of 40%.

	CFD	**EXP**	**% Cd Change CFD**	**% Cd Change EXP**
Base Model	0.270	0.250		
Case 1	0.260	0.220	4%	12%
Case 2	0.159	0.130	41%	48%
Case 3	0.167	0.150	38%	40%

Table 1 C_d comparison between CFD & Experiments

Flow Field in the Near Wake Region

The flow separation behind the body results in the formation two large vortices in the wake as shown in Figures 7-10 for the base model and other three cases. For the base model case as shown in Figure 7 that the two vortices are similar in size while the predicted bottom vortex is much smaller than the top one. Nevertheless the general features of the predicted top vortex is similar to those of the experimental one (length and centre of the vortex). The agreement between the predicted two vortices and the experimental ones is better for case 1 than that for the base model case in terms of the general features of those two vortices (similar vortex length and location of the vortex centre) although the predicted bottom vortex is still slightly smaller. However, for case 2 the predicted length is quite different to that of the experimental one, about 30% over-prediction of the length. This over-prediction of the vortex length is still visible in case 3 with about 20% over-prediction.

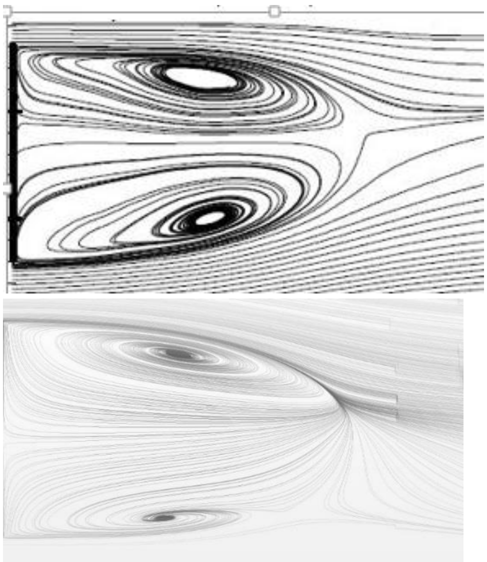

Figure 7 Wake streamlines showing two large vortices for the base model: top – Experiment, bottom- CFD.

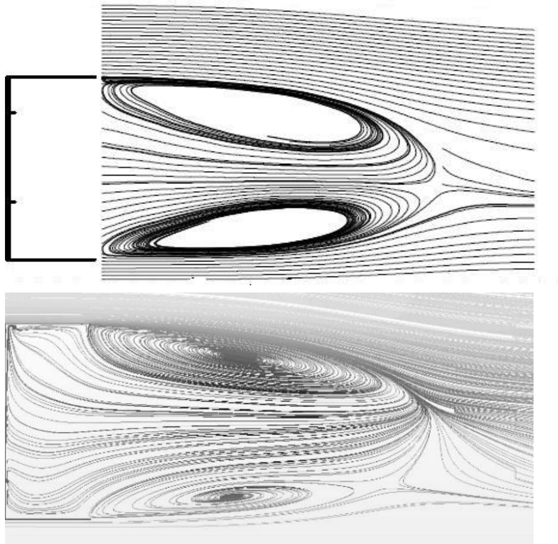

Figure 8 Wake streamlines showing two large vortices for case 1: top – Experiment, bottom- CFD.

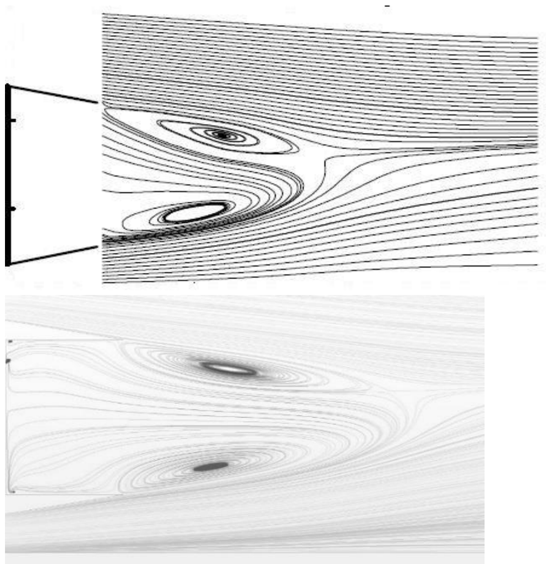

Figure 9 Wake streamlines showing two large vortices for case 2: top – Experiment, bottom- CFD.

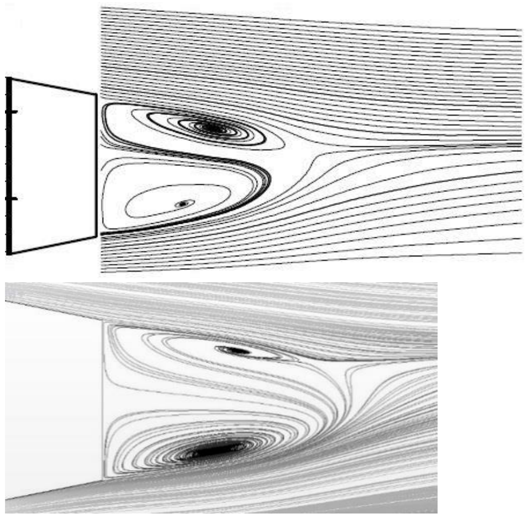

Figure 10 Wake streamlines showing two large vortices for case 3: top – Experiment, bottom- CFD.

Figures 11 and 12 below shows the normalised mean axial velocity profile in the vertical centre plane for the four cases at the axial location 1.5H downstream of the back plate of the model. It can be seen that a good agreement, both in terms of profile shape and magnitude, has been obtained between the predictions and the experimental data for all cases, especially for case 2 with a very good agreement. It can be seen that the recirculation region length (axial direction) and width (vertical direction are different for different cases.

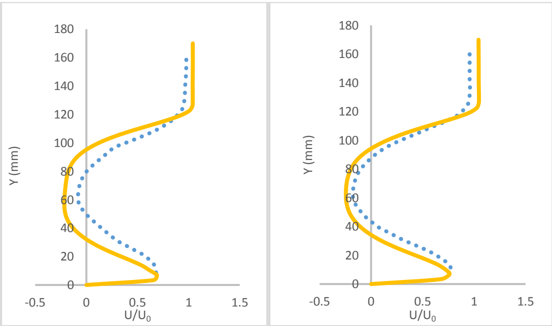

Figure 11 Mean velocity profile: left- base case, right - case 1

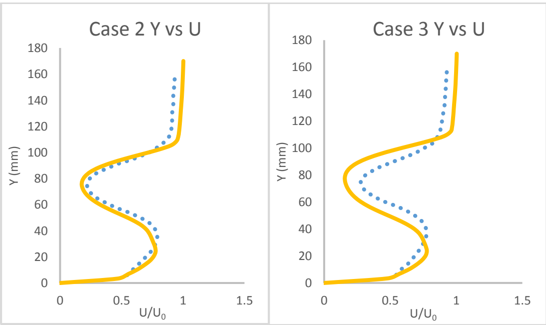

Figure 12 Mean velocity profile: left - case 2, right - case 3

The over-prediction of the recirculating region was also observed by Rumsey *et al*. [12] in their CFD study. Their predicted reattachment location is significantly further downstream than the location documented in the experiments. They proposed that this was due to the under-prediction of turbulent shear stresses, leading to less turbulent mixing inside the separated region which results in delayed reattachment.

Figures 13 and 14 show the normalised Reynolds stress u'^2 at the same location and it can be seen from the experimental data that there are two distinct peaks for all cases, which are well captured by the predictions although the lower peak magnitude is over-predicted for base case and case 1. Nevertheless the overall agreement between the predictions and the experimental data is quite good. It can also be seen that for cases 2 and 3 (especially case 2) that turbulent kinetic energy is significantly reduced. The significant drag reduction for cases 2 and 3 may partly be due to the lower turbulent kinetic energy levels in those two cases. Similarly an overall good agreement between the predicted shear stress and the experimental data is obtained as shown in Figures 15 and 16, especially with a very good agreement for case 2.

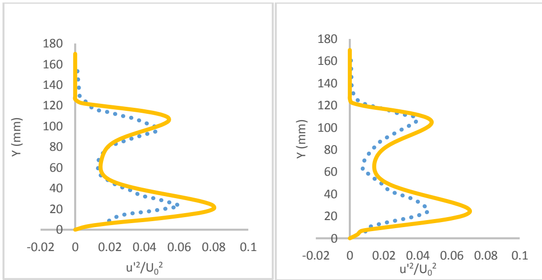

Figure 13 u'^2 profile: left- base case, right - case 1

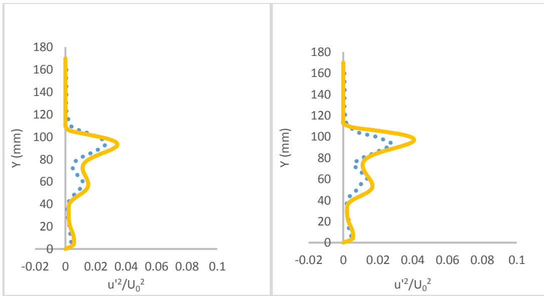

Figure 14 u'^2 profile: left - case 2, right - case 3

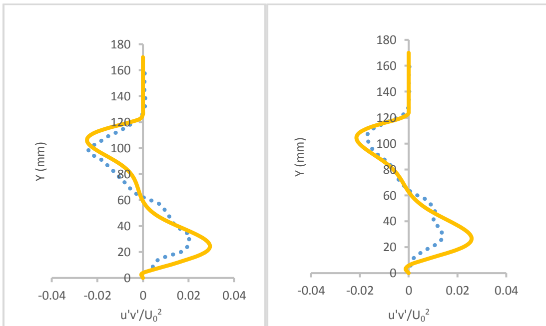

Figure 15 u'v' profile: left- base case, right - case 1

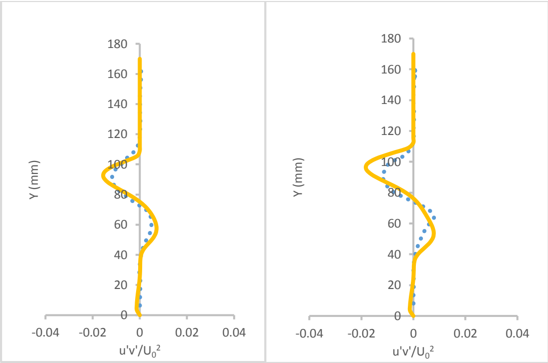

Figure 16 u'v' profile: left - case 2, right - case 3

DISCUSSION & CONCLUSION

This paper describes a numerical study of flow over a bluff body with drag reduction devices using the RANS approach. Three turbulence models (realizable *k-ε*, *SST k-ω* and RSM) have been utilized and their performance has been assessed. As expected that the RSM model provides the best results.

Four cases (one without drag reduction device denoted as base model case and three cases with devices) have been considered in the present study. The experiments showed that adding a cavity to the back of the model (case 1) reduced drag by 12% while the predicted reduction was 4%. The predicted drag reductions for the other two cases are closer to those of the experimental values, for case 2 (boat-tail with cavity) it is 48% (exp) against 41% (CFD) and for case 3 (boat-tail without cavity) it is 40% (exp) against 38% (CFD), a very good agreement for case 3 indeed.

The near wake flow field is analysed and the gross features of two distinct vortices are reasonably well captured by the predictions. The predicted vortex length for the base model case and case 1 compare reasonably well with the experimental one. However, the predicted length for case 2 is about 30% longer than the experimental one and for case 3 it is 20% longer.

The predicted mean axial velocity profiles, u'^2 and u'v' components of the Reynolds stresses at a distance 1.5H from the back of the model agree reasonably well with the experimental data for all four cases. The recirculation length and the peaks in the u'^2 and u'v' profiles are all well captured by the predictions despite the peak magnitude is over-predicted for base case and case 1. Nevertheless the overall agreement between the predictions and the experimental data is good, demonstrating that for this type of flow with its inherent unsteadiness, the RANS approach can produce reasonably good results with significant save in computational cost.

REFERENCE

[1] Bearman, P. (2009). Bluff body flow research with application to road vehicles. See Browand et al. 2009, pp. 3–13.

[2] Ortega, J.M., Salari, K. (2004). An experimental study of drag reduction devices for a trailer underbody and base. Presented at AIAA Fluid Dyn. Conf. Exhib., 34th, Portland, OR, AIAA Pap. 2004-2252.

[3] Bradley, R. (2000). Technology roadmap for the 21st century truck program. Tech. Rep. 21CT-001, US Dep. Energy, Washington, DC

[4] Wood, R.M. (2006). A discussion of a heavy truck advanced aerodynamic trailer system. Presented at Int. Symp. Heavy Veh. Weights Dimens, University Park, PA.

[5] Ahmed, S.R., Ramm, G., Faitin, G. (1984). Some Salient Features of the Time – Averaged Ground Vehicle Wake. SAE-TP-840300.

[6] Bayraktar, L., Landman, D. & Baysal, O. (2001) Experimental and computational investigation of 'Ahmed body' for ground vehicle aerodynamics. SAE 2001 Transactions Journal of Commercial Vehicles-V110-2.

[7] Duell, E. G. & George, A. (1993). Measurements in the unsteady near wakes of ground vehicle bodies. SAE Technical Paper 930298, 1–8.

[8] Khalighi, B., Balkanyi, S.R., Bernal, L.P. (2013). Experimental investigation of aerodynamic flow over a bluff body in ground proximity with drag reduction devices. Int. J. Aerodynamics 3:217–233.

[9] Khalighi, B., Chen, K.-H. & Iaccarino, G. (2012). Unsteady aerodynamic flow investigation around a simplified square-back road vehicle with drag reduction devices. J. Fluids Engng 134 (6).

[10] Krajnovic, S. & Davidson, L. (2001). Large-eddy simulation of the flow around a ground vehicle body. SAE Paper.

[11] Krajnovic, S. & Davidson, L. (2004). Large-eddy simulation of the flow around simplified car model. SAE paper pp. 01–0227.

[12] Rumsey, C.L., Gatski, T.B., Sellers, W.L., Vatsa, V.N. and S.A. Viken, S.A. (2005). Summary of the 2004 CFD Validation Workshop on Synthetic Jets and Turbulent Separation Control, NASA Langley Research Centre, AIAA 2005-1270.