Wind and Structures, Vol. 5, No. 2~4 (2002) 359-368
DOI: http://dx.doi.org/10.12989/was.2002.5.2_4.359

Bluff body asymmetric flow phenomenon – real effect or solver artefact?

Tanya Prevezer† and Jeremy Holding‡

Aerodynamics Team, AEA Technology Rail, rtc Business Park, London Road, Derby, DE24 8YB, UK

Adrian Gaylard‡ and Robert Palin‡

Fluids Group, MIRA, Watling Street, Nuneaton, UK

Abstract. This paper describes a CFD investigation into the flow over the cab of a bluff-fronted lorry. Several different simulations were undertaken, using the commercial codes: CFX, Fluent and PowerFLOW. Using the k - ε turbulence model, the flow over the cab was symmetric, however, using more accurate turbulence models such as the RNG k - ε model or the Reynolds Stress Model, the flow was asymmetric. The paper discusses whether this phenomenon is a real effect or whether it is a solver artefact and the study is supported by experimental evidence. The findings are preliminary, but suggest that it has a physical origin and that it may be aspect ratio-dependent.

Key words: CFD; CFX; PowerFLOW; asymmetry; turbulence model; aspect ratio; unstable; bluff.

1. Introduction

During an EPSRC-funded investigation into transient slipstream effects generated by a bluff lorry (Baker *et al.* 2000), a CFD simulation of the vehicle was formulated to provide additional flow visualisation. Whilst the lorry geometry was simple, the results were not as anticipated: an asymmetric flow was predicted over the cab. Several modifications were made to the simulation to see if a more expected flow pattern would result. These included using a variety of turbulence models, mesh topologies, different methods of discretisation, and even changing the modeller and the CFD code. The flow remained asymmetric, whichever combination of parameters was used.

This paper outlines the details of the study and presents preliminary results. Several questions are posed by the surprising nature of the asymmetric results and these are discussed in turn, with further supporting modelling and experimental evidence. The objective of the paper is to provoke discussion as to whether the asymmetry is physically present and the validity of assumptions commonly made in CFD practise by presenting the results of an apparently simple CFD model.

---

† Ph. D
‡ Mr.

2. Geometry

The lorry in this model was based on the dimensions of a model-scale lorry, which was tested in AEA Technology Rail's unique Moving Model Rig and wind tunnel. A schematic representation is shown in Fig. 1. The lorry was modelled without a symmetry plane.

3. CFD modelling

3.1. RANS-based solvers

The initial CFD modelling was carried out using CFX 5.3. This is a Reynolds Averaged Navier Stokes (RANS) based solver, which uses a non-structured mesh, based on tetrahedral element discretisation. Near the boundaries, it has the capability of 'inflating' surface triangular elements into structured prismatic elements, which can be used to improve boundary layer representation. CFX 5 uses a coupled solver, whereby all terms are solved together.

The discretisation scheme used by CFX 5 is a conventional upwind differencing scheme with numerical advection correction (NAC) for the advection terms in the governing equations. The scheme can be run with a 'blend' of 1st and 2nd order discretisations, where the user can specify a desired proportion of the 2nd order correction.

The lorry model was situated in the centre of a fluid domain of dimensions 6500 mm $\times$ 915 mm $\times$ 500 mm, which is approximately six lorry lengths upstream and downstream of the lorry; about 3.5 lorry widths either side of the lorry and about 3.5 lorry heights above the lorry. The lorry was modelled 5 mm above a moving ground plane.

Further to the use of CFX, the lorry was also modelled in Fluent (a similar standard Reynolds Averaged Navier Stokes (RANS) solver) and in EXA Digital Physics PowerFLOW, a more unusual CFD solver.

3.2. PowerFLOW

The PowerFLOW software has created a large degree of interest within the automotive aerodynamics community since its launch in 1996. Its accuracy is said to match or exceed results obtained from

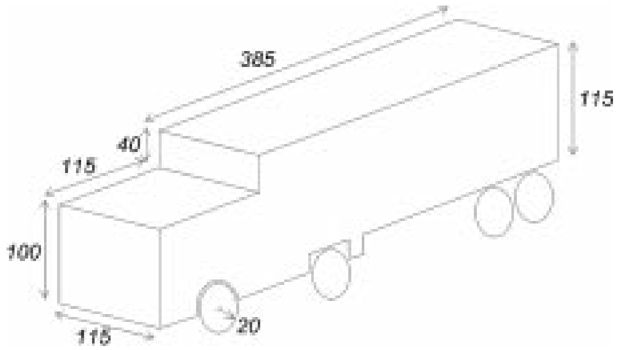

Fig. 1 Schematic representation of the lorry (Dimensions in mm. Not to scale)

the more traditional RANS solvers. It is based on an extension of lattice gas theory to high Reynolds number flows. The detail of this theoretical approach has been extensively discussed in the literature and Anagost *et al.* (1997) provide both a review of this along with an analysis of a simple external aerodynamics “benchmark”. In recent years, workers have reported an increasing number of vehicle flowfields simulated with a degree of accuracy matching or exceeding those obtained using currently available RANS approaches. (See Gaylard *et al.* 1998, Perzon *et al.* 1998 and Lietz *et al.* 2000) In general terms, this method has the characteristics listed below:

- Discrete particle method, where the fluid is represented by “particles” distributed through a lattice, all having discrete velocities and locations.
- Volume discretisation via a cubic lattice, which for a vehicle model typically contains fifteen to thirty million three-dimensional lattice cells (“voxels”) to resolve the flow domain.
- The particle distribution dynamics via the application of discrete kinetic theory provides for full recovery of the Navier-Stokes continuum fluid equations for the behaviour of the macroscopic fluid properties.
- The vehicle surface is defined by two-dimensional surface elements (“surfels”), which cut the near-surface voxels to define the surface/fluid interface.
- The methodology is inherently transient.
- Unresolved boundary layer is represented via a pressure-sensitised “law-of-the-wall” (i.e., wall function) approach.
- For external flows, PowerFLOW uses, in effect, a constant eddy viscosity turbulence model, and a near-wall turbulence model for boundary layers. The use of a high-resolution lattice enables this simple turbulence modelling approach to deliver an acceptable simulation.

The parametric space explored is shown in Table 1, below.

Table 1 CFD simulations undertaken

Simulation No.	Code	Mesh	Turbulence model	Discretisation scheme
1	CFX 5.3	tetrahedra, prismatic boundary elements	k - ε	1st order
2	CFX 5.3	tetrahedra, prismatic boundary elements	RNG k - ε	1st order
3	CFX 5.3	tetrahedra, prismatic boundary elements	RNG k - ε	‘blend factor’ of 0.75
4	CFX 5.3	tetrahedra, prismatic boundary elements	RSM	‘blend factor’ of 0.75
5	CFX 5.3	hexahedra	RSM	‘blend factor’ of 0.75
6	CFX 4.2	regular lattice	k - ε	1st order
7	Fluent	all tetrahedra	RNG k - ε	‘high order’
8	Power FLOW	regular cubic lattice, local embedded refinement	constant eddy viscosity+near-wall	–

4. Results

The models revealed the following general flow features:

- a large separation over the cab, which recirculated over the cab
- a very small separation over the trailer

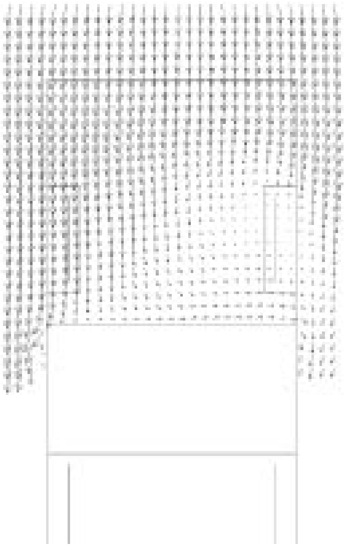

Fig. 2 Output from CFX showing asymmetry across lorry cab. (Flow enters from the top; arrows represent velocity vectors; flow is displayed on a plane parallel with the ground, above the cab surface.)

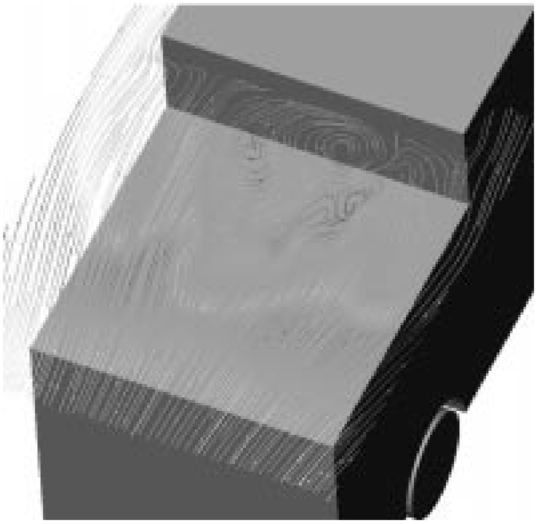

Fig. 3 Output from PowerFLOW showing asymmetric streamlines above the cab

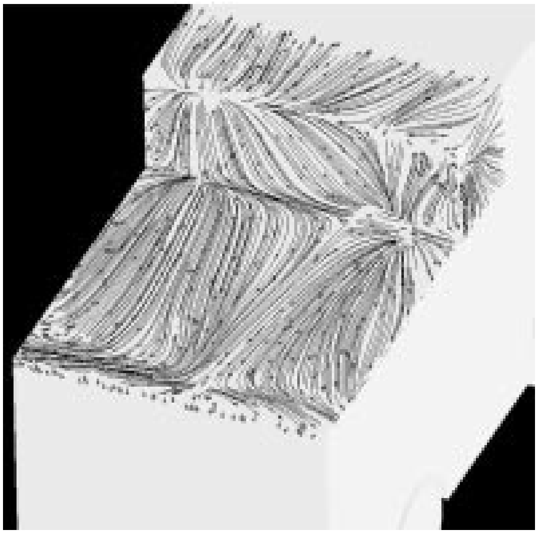

Fig. 4 Output from PowerFLOW showing asymmetric surface flow on the cab and trailer front

- well pronounced wake vortices
- some separation round the sides of the cab

In addition to these general features, the following phenomenon over the cab was noticed :

- Using the k - ε model, the solution showed a symmetrical solution
- Using the RNG k - ε model and a RSM, the solution showed an asymmetric solution, whereby the flow preferentially veered to one side of the lorry over the cab and there was a region of low pressure over the front corner of the cab. (See Fig. 2) The asymmetry was more pronounced with the RSM.
- When the lorry was remodelled in Fluent and in PowerFLOW, the asymmetry was also evident. (See Fig. 3 and Fig. 4)

5. Discussion

The results of this investigation pose an interesting question: "Is this asymmetry a real effect, or a simulation artefact?" All instincts point towards the latter. However, the probability of the same artefact being present in simulations made with three different CFD packages, each independently set up by different users, would seem to be low. This is compounded by the use of two very different CFD solver schemes, namely RANS and LGD. Hence, it may also be prudent to raise the question "Is there a physical cause for the asymmetry?"

5.1. Literature review

The results from these CFD simulations were not as expected, since it had been assumed that the flow would be symmetrical, at least at the front of the lorry. In a literature search of similar bluff shapes, no mention of asymmetric flows was found, but it is suspected that, in the majority of these simulations, a symmetry plane was modelled, so any potential asymmetries may not have been noticed. The decision not to use a symmetry plane in *this* investigation was made to enable any wake asymmetry to be studied, however any other asymmetries were not envisaged.

The only paper found to mention any unusual results with a truck or bluff body were from Perzon *et al*. (1999), which reports on RANS (STAR-CD) modelling and wind-tunnel measurements on a simplified Volvo FH truck. Whilst this vehicle is very different to the lorry model investigated in this paper, the authors reported that all the turbulence models they investigated failed at the transition between the tractor and trailer. This numerical error was seen to convect downstream. The simplified truck had a full-faired transition between tractor and trailer, yet the following possible causes of error were suggested:

- poor mesh resolution in this zone;
- change in tunnel blockage

The authors stated that they did not fully understand the effect. The model reported in *this* paper has a radical transition between the cab and the trailer and could therefore be more susceptible to this type of error.

5.2. Steady state or transient?

The second consideration is whether or not the model can be solved as a steady-state problem or whether transient analysis is essential. A steady-state solution of a transient problem should produce, if not a time-averaged solution, at least an indication of what a time-averaged solution would be. The CFX and Fluent models were solved steady-state, but PowerFLOW is inherently transient and the PowerFLOW results are time-averaged over a significant sampling period. Since the results were similar for both solutions, this seems not to be the cause of the asymmetry.

5.3. Turbulence models

The effect produced by using different turbulence models provokes further contemplation. It is known that the k - ε model may lead to numerical diffusion, causing significant errors particularly where separation and recirculation are involved. The RNG k - ε model and more so the RSM are thought to provide more accurate solutions and more flow details (e.g., Easom *et al.* 1998). With this in mind, and the asymmetry of the flow increasing as the turbulence model increased in accuracy, the question may be posed "*What turbulence model should be used to give satisfactory results*?" Perhaps an over-prediction of kinetic energy in the recirculation bubble, by the k - ε model, obscured any potential asymmetries, which were then captured by the other turbulence models. This may also have damped out any time-varying flow features.

5.4. Grid resolution

A further source of error in the CFX model could have been due to poor grid resolution. There were approximately 2×10^5 elements. This was not, however, the case for PowerFLOW where the number of "voxels" exceeded 15 million, so it is unlikely that this caused the asymmetry. The PowerFLOW mesh is also symmetric about the vehicle axis, being automatically generated.

5.5. Aspect ratio

To complement the full lorry model, further CFD simulations were made using a simplified representation of the lorry as two blocks. These were modelled with different aspect ratios (height of cab : trailer), but all other aspects of the simulations were identical. The two aspect ratios modelled can be seen in Fig. 5. With the RNG k - ε model, the solution of the blocks, which had the same aspect ratio as the lorry, gave asymmetric results, as before. However, the solution of the blocks with the other aspect ratio showed *symmetric* results. This suggests that perhaps the solution was aspect ratio-dependent.

5.6. Reynolds Number

Reynolds Number effects were also considered as a potential source of error, but having run the simulation at more than one speed, only to achieve similar results, this too was discounted as the source of the asymmetry. This is perhaps not surprising, as changes due to boundary layer transition will not be modelled by this type of RANS solver, or indeed PowerFLOW.

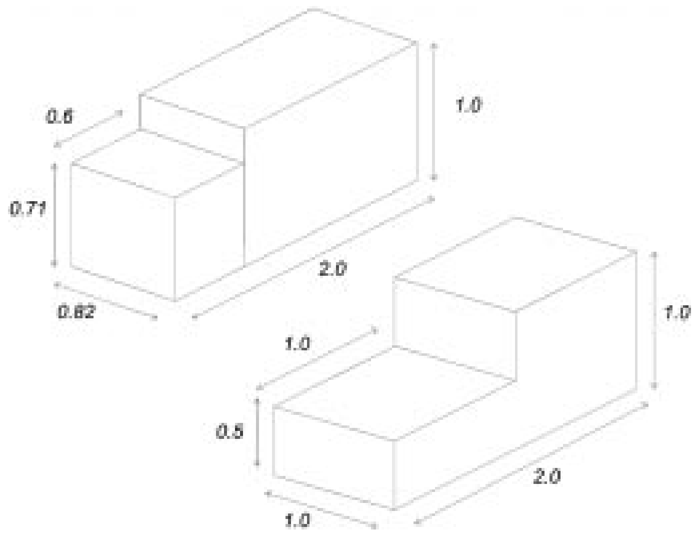

Fig. 5 Two different aspect ratios

6. Experimental evidence

With the question of whether or not the asymmetry was real, still unanswered, the model was tested in the MIRA model-scale wind tunnel (Brown *et al.* 1998). The model was mounted on pins 5 mm above a fixed ground with floor boundary layer suction. Here the model was examined using three flow-visualisation techniques :

- cotton tufts
- smoke (at 20 m/s)
- fluorescene surface visualisation (at 30 m/s over a 4 minute period)

Force data was also taken at 30 m/s. Somewhat surprisingly, the experiment seemed to confirm the CFD: there was definitely evidence of asymmetry. Fig. 6 shows smoke visualisation, where the velocity of the smoke was very small compared with the velocity of the flow. It can be clearly seen that the smoke is drawn diagonally across the cab. Over time, the smoke nearly always flowed diagonally to the left of the cab, but occasionally it chose to flow to the right. Indeed, the deposition of liquid residue left by the smoke was thicker on the left of the cab. Fig. 7 shows the fluorescene surface visualisation. The dark areas are regions of fast flow, where the fluorescene has been scoured. The lighter areas are areas of slow flow where there has been deposition. Again, asymmetry is evident. This is particularly noticeable on the vertical surface that joins the cab to the trailer. It was more difficult to see any asymmetry of the flow when the cotton tufts were used. This was due to the tufts flickering rapidly in all directions. This did, however, indicate that the flow was highly unsteady.

These results were not anticipated, nevertheless, the tests clearly showed asymmetry. Perhaps the CFD was correctly predicting the flow after all.

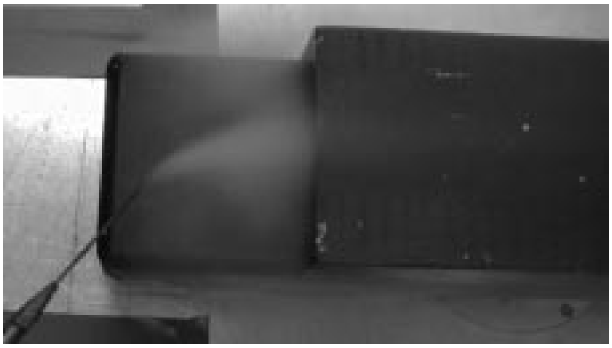

Fig. 6 Flow visualisation using smoke

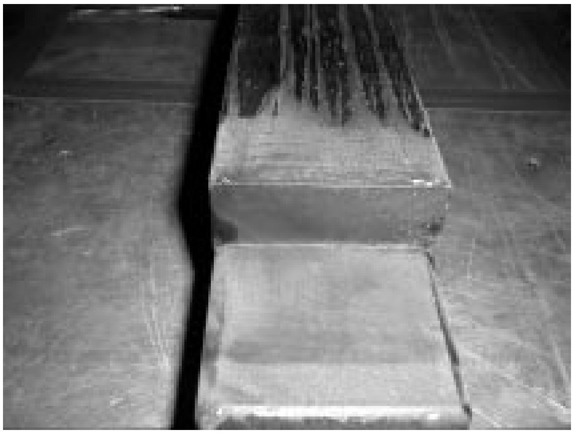

Fig. 7 Surface flow-visualisation using fluorescene

7. Conclusions

7.1. General

The aim of this paper has been to provoke debate, particularly in the aerodynamics simulation community, by presenting an apparently simple bluff body that has produced simulation and experimental results that call into question assumptions that are frequently made in CFD practise.

Although this investigation must be considered to be at a preliminary stage, the evidence presented argues in favour of the flow asymmetry being a physical effect rather than a solver artefact. Therefore, it is that appropriate to offer some comment on possible underlying causes and cautionary advice to CFD practitioners.

7.2. Possible flow structure

From watching the flow-visualisation, it was clear that the flow was unsteady. This was also reflected in the scatter of lift force measurements. The PowerFLOW results, which were processed every 10 milliseconds, also showed general unsteadiness

With this in mind, it could be argued "*Why should we expect the time averaged flow to be symmetric*?". It would generally be expected that the flow around a bluff body may exhibit self-induced unsteadiness. This, in turn, could be seen in the context of one or more general flow structures. The most likely explanation is probably that the flow is actually bistable and that the symmetric solution is the 'artefact' which in practice does not persist. In this case the flow may be behaving in a similar manner to that observed with fluidic switching and choosing to flow in one direction preferentially, due to an imperceptibly small change in turbulence, onset flow angularity, or some other perturbation. The displacement surface may then grow thicker on one side of the body than the other, thus presenting an apparently asymmetric body to the oncoming flow. This may then cause the flow to select one direction preferentially.

More detailed flow measurements and simulations would be required to substantiate this hypothesis. In particular, examining the effect of small onset flow yaw angles may provide some further insight.

7.3. CFD implications

This work adds to the growing body of opinion among CFD practitioners in the automotive field that the imposition of centre-line symmetry, whilst tempting for pragmatic considerations of resources and simulation turn-around time, is a potential source of significant error. (See Gaylard *et al.* 1998 and Perzon *et al.* 1998). This body of work has typically focussed on errors in wake prediction due to the imposition of a symmetric wake and the exclusion of lateral energy transfer from periodic vortex shedding. This literature also provides some evidence of the deleterious effect of this assumption in modelling automotive geometries with large recirculation features prior to the wake (i.e., "notchback" saloons). In the work reported in this paper, an asymmetric (in both an instantaneous and time-averaged sense) flow structure has been identified both numerically and via experiment that would preclude the use of the centre-line symmetry assumption in CFD simulations for this geometry.

The question of how generally applicable this observation is has still to be answered. There is some evidence that the effect is aspect ratio-dependant. Therefore this issue may not arise with all geometries of this general type. Hence, it is not possible at this stage to comment on the implications for CFD simulation of other bluff bodies, such as buildings. However, the work presented in this paper argues for caution in assuming flow symmetry.

It is noteworthy that the CFD approaches used managed to predict an unexpected solution which has subsequently been supported by experimental evidence, *provided that the standard* k - ε *turbulence model was not used*. The prediction was made before the experiment was undertaken, which historically has not been the usual order of events, therefore encouragingly, the CFD modelling has been seen to be a useful predictive tool. In addition, steady state CFD simulations with relatively simple numerical models provided an indication of the structure of this complex flow signalling the need for further investigation. This is also encouraging for CFD practitioners as steady state flow is one of the most commonly made assumptions in the practise of CFD.

In the light of these results, CFD practitioners, would also be well advised not to dismiss unexpected results from CFD simulations with undue haste.

In summary, this paper has identified a curious phenomenon, associated with flow over a bluff body. The asymmetry seen in CFD results was not expected and was initially thought to be an artefact associated with the solver. The asymmetry was borne out by experimental evidence, which gives weight to the conclusion that the flow may well be asymmetric. In any event the study has provided an interesting case, which merits further investigation, particularly with regards to aspect ratio dependency, elucidation of the flow structure and its cause.

Acknowledgements

The authors would like to thank the following for their input into this investigation:

AEA Technology Rail, MIRA, EPSRC, John Keeton (AEA Technology Engineering Software), Nigel Wright (University of Nottingham), Geoff Le Good (Rolls Royce and Bentley Motor Cars) and Roger Gawthorpe.

References

Anagost, A. Alajbegovic, A., Chen, H., Hill, D., Teixeira, C. and Molvig, K. (1997), “Digital physics analysis of the novel body in ground proximity”, *SAE Int. Congress and Exposition*, Detroit, Michigan, USA. SAE Paper No. 970139.

Baker, C.J., Dalley, S. and Johnson, T. (2000), “Measurements of the slipstream and wake of a model lorry”, To be published at *3rd MIRA Int. Conference on Vehicle Aerodynamics*, Rugby, UK.

Brown, M., Baxendale A. and Hickman, D. (1998), “Recent enhancements of the MIRA model wind-tunnel”, *2nd MIRA Int. Conference on Vehicle Aerodynamics*, Coventry, UK.

Easom, G.J., Wright, N.G. and Hoxey, R.P. (1998), “Improved computational models for wind engineers”, *Proc. 4th UK Conference on Wind Engineering*, Bristol.

Gaylard, A.P., Bickerton, J. and Howell, J.P. (1998), “Current issues in the use of CFD to predict aerodynamic characteristics of car shapes”, *2nd MIRA Int. Conference on Vehicle Aerodynamics*, UK.

Lietz, R., Pien, W. and Remondi, S. (1998), “A CFD validation study for automotive aerodynamics”, *SAE Int. Congress and Exposition, Detroit*, Michigan, USA. SAE Paper No. 2000-01-0129.

Perzon, S., Sjõrgren, T. and Jõnson, A. (1998), “Accuracy in computational aerodynamics Part 2: base pressure”, SAE Int. Congress and Exposition, Detroit, Michigan, USA. SAE Paper No. 980038.

Perzon, S., Janson, J. and Höglin, L. (1999), “On comparisons between CFD methods and wind tunnel tests on a bluff body”, SAE Int. Congress and Exposition, Detroit, Michigan, USA. SAE Paper No. 1999-01-0805.