

Control of the aerodynamic drag of Ahmed model with slot jet

Wang Hanfeng^a, Xu Mengxia^a

^aNational engineering laboratory for high-speed railway construction, Central South University,
Changsha, China

ABSTRACT: The effects of a slot jet on the aerodynamic drag of an Ahmed model with the slant angle of 25° is investigated using LES technique. Three jet directions are considered, i.e. along slant surface (case 1), backwards horizontally (case 2) and 30° inclined upward (case 3). There is a pair of symmetrical streamwise tail vortices, which accounts for a major part of the aerodynamic drag. The slot jet has significant effects on the near wake and aerodynamic drag. For case 1, the tail vortices are enhanced and the drag is increased by about 0.91%. While for cases 2 and 3, the application of slot jet extends the reverse flow zone and suppresses the tail vortices. The total aerodynamic drag reduces by 7.88% and 3.64% for these two cases, respectively.

KEYWORDS: Ahmed model, Tail vortex, Flow control, Drag reduction, LES.

1 INTRODUCTION

Control of the aerodynamic drag of a road vehicle is one of the most important research problems about vehicle aerodynamics, which is directly related to the fuel efficiency. Generally, the drag of a vehicle is proportion to the square of its speed. The aerodynamic drag on a road vehicle typically accounts for about 70% of its total resistance when its speed reaches 90 Km/h. The control of aerodynamic drag becomes widely concerned because of the skyrocketing fuel cost and increasing environmental burden from fossil fuels.

The geometrical figure of road vehicle is usually a three-dimensional (3D) bluff body. The aerodynamic drag on a bluff body is largely determined by the pressure difference between its front and rear end^[1,2], and closely related to the flow separation near the rear end^[1]. The flow in the near wake of this 3D bluff body is highly complex, unsteady. To facilitate the comparative aerodynamic studies, Ahmed^[3] proposed a generic vehicle configuration, named "Ahmed model". Despite that the Ahmed model has been proposed for nearly three decades, it is still frequently used for the investigation of the 3D wake flow and aerodynamic performance for road vehicles^[4-7]. The near wake structure and consequently the aerodynamic drag of Ahmed model strongly depend on the rear slant angle^[2,3]. For the slant angle of approximately 25° to 30° , large longitudinal vortex structures occur in the near wake, which lead to a considerable increase in aerodynamic drag.

Recently, the flow control techniques have been more and more studied for drag reduction, with the Ahmed model as a benchmark^[7]. Some passive control methods, such as: tail clapboards^[8], flaps near the flow separation point^[9] and deflector^[10] etc., have been successfully used for drag reduction. However, most of these passive control techniques introduce quite unsightly appendages on the body, which is in contradiction with design constraints^[7]. This fact makes active control by jet and/or suction very attractive. For Ahmed model with the slant angle of 25° , Roumeas *et al.*^[11] used steady aspiration on the top edge of the slant and obtained about 17% drag reduction. Krajnovic *et al.*^[12] made numerical simulations of the same case and obtained a little bit more than 7% reduction using steady blowing and suction at the slant top edge.

In the present paper, the flow around and the aerodynamic drag on an Ahmed model with a slant angle of 25° are investigated using large eddy simulation (LES) technique. A slot jet at

slant top edge is employed to control the flow. Three jet directions are presently studied to examine the effects of jet direction on the near wake flow structure and aerodynamic drag.

2 LES & CONTROL CASES

2.1 LES method

The governing equations of LES are the incompressible Navier-Stokes and the continuity equations filtered with the implicit spatial filter of characteristic width Δ , which is the grid resolution. The goal of this filtering is to decompose the fluid motion into a large-scale component that is resolved directly, and the small subgrid scale (SGS) which is modeled using SGS model. The filtered LES equations are shown in Eqs. (1 & 2), where the overbar denotes the filtered variable, ρ and ν are the density and kinematics viscosity of fluid, respectively.

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_i \partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} \quad (2)$$

The influence of the small scale turbulence on the large energy carrying scales in Eq. (2) appears in the SGS tensor, $\tau_{ij} = \overline{u_i u_j} - \bar{u}_i \bar{u}_j$ [4]. The algebraic eddy viscosity model proposed by Smagorinsky [13], shown in Eq. (3), is presently used for its simplicity and low computational cost, where the SGS viscosity $\nu_t = (C_s \Delta)^2 |\bar{S}|$. $|\bar{S}|$ is the resolved rate-of-strain tensor, and $|\bar{S}| = (2\bar{S}_{ij}\bar{S}_{ij})^{0.5}$, where $\bar{S}_{ij} = \frac{1}{2}(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i})$. The filter width Δ is defined as $\Delta = (\Delta_x \Delta_y \Delta_z)^{1/3}$, where Δ_i are the computational cell sizes in three coordination directions. Readers may refer to Krajnovic & Davidson [4] for detailed description of the LES method presently used.

$$\tau_{ij} = \overline{u_i u_j} - \bar{u}_i \bar{u}_j = -2\nu_t \bar{S}_{ij} + \frac{\tau_{kk} \delta_{ij}}{3} \quad (3)$$

2.2 Generic vehicle mode and numerical details

An Ahmed model with the slant angle of 25° is presently investigated. The dimensions are shown in figure 1(a), the length (l), width (w) and height (h) are 1044 mm, 389 mm and 288 mm, respectively. The computational domain is $8l$ in length, $10w$ in width and $5h$ in height. The model is $2l$ from the inlet and 50mm above the bottom wall. The blockage ratio is 2%. Definition of the coordinate system is shown in figure 1(b).

The computational domain is meshed into hexahedrons. The grid size reduces gradually with the wall approached, so that most of the grids concentrate around the Ahmed model. The total number of grid is about 180 million. The y^+ on the model surface ranges from 0.7 to 33, which is considered enough to capture the larger scale flow structure.

A uniform velocity U_∞ of 25m/s is used as the inlet boundary layer condition in present LES. Corresponding Reynolds number, based on U_∞ and l , is 1.75×10^6 . Pressure boundary condition is used at the exit of computation domain. The lateral and ceiling surfaces of the computa-

tional domain are treated as slip surfaces using symmetry condition. No slip boundary layer conditions are used on the surfaces of Ahmed model and ground wall.

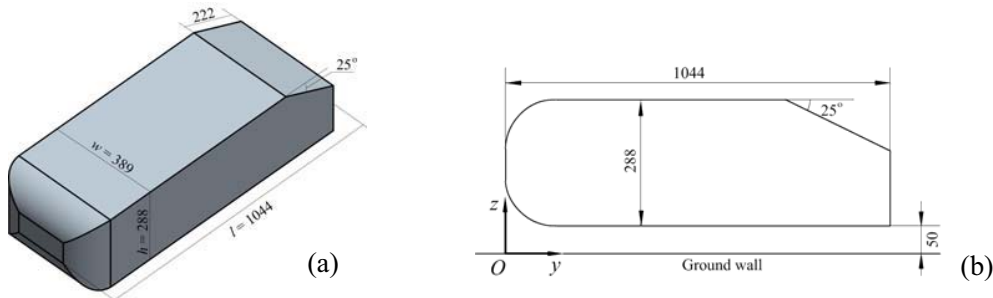


Figure 1. Generic vehicle model and control cases: (a) dimension of Ahmed model, (b) definition of the coordinate system. All dimensions are in mm.

2.3 Control cases

To manipulate the near wake flow and subsequently the aerodynamic drag on the Ahmed model, a slot jet at the top edge of the slant surface was used. Three different jet directions were tested. For case 1, the direction of the slot jet is along the slant surface, for case 2, the jet is horizontally backwards, while for case 3, the jet direction is inclined upwards with an angle of 30° , as shown in figure 2. To provide a benchmark, the case without control was taken as case 0. The length of the slot was equal to the width of the model, and the width of slot is 11 mm, 5% of the slant length. The slot jet velocity (U_j) is 9 m/s for the three cases, corresponding to a jet intensity $c_\mu = 5 \times 10^{-3}$. c_μ is defined by Eq. (4)^[14], in which A_J and A_B are the projected area of the slot jet and the Ahmed body in flow direction, respectively.

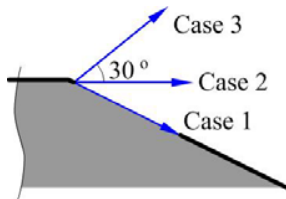


Figure 2. Configuration of control cases.

$$c_\mu = \frac{A_J}{A_B} \cdot \frac{U_j^2}{U_\infty^2} \quad (4)$$

3 RESULTS & DISCUSSIONS

3.1 The new wake flow structures

Figure 3 presents the isosurfaces of instantaneous streamwise vorticity, i.e., ω_y^* . In present paper, the superscript asterisk represents the normalization with U_∞ and/or l . Obviously, the near wake is highly complex and 3D. A pair of streamwise tail vortices occurs near the side edges of slant face (as shown in figure 3), similar to that observed by Vino *et al.*^[15] with smoke visualization technique. The time-averaged near wake structure is shown in figure 4, together with the schematic near wake structure proposed by Ahmed *et al.*^[3] for comparison. The shear layers from the ceiling and bottom surfaces form a recirculation zone downstream the body. A pair of symmetrical tail vortices dominates the near wake, which is responsible for a significant part of

the lift and drag^[16]. The time averaged flow structure of present LES, as shown in figure 4(b), corresponds well with the proposed flow model, which validates the present results. For an Ahmed model with the slant angle of 25°, the reported drag coefficient (C_D) was scattered for different techniques, e.g. 0.30 by Ahmed et al.^[3] and 0.32 by Vino et al.^[15] The present C_D for case 0 is 0.33, which will be used as a benchmark to evaluate the effectiveness of the slot jet on drag reduction.

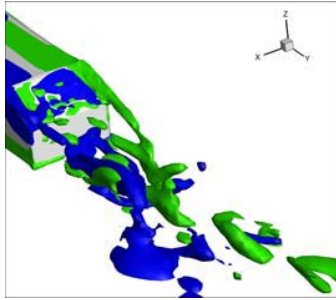


Figure 3. Isosurfaces of instantaneous streamwise vorticity, $|\omega_y^*| = \pm 6$.

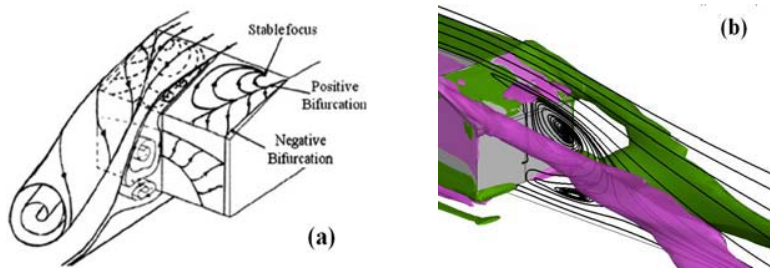


Figure 4. Time-averaged near wake structure: (a) Sketch of flow structure^[3], (b) Present LES result.

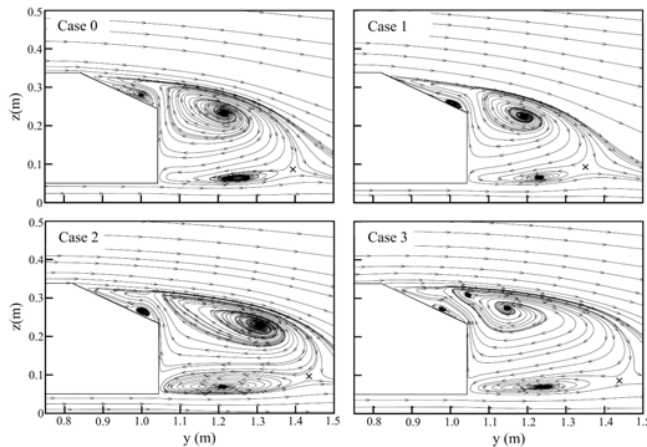


Figure 5. Time-averaged sectional streamlines in the central symmetry plane for different cases.

The time-averaged sectional streamlines in the central symmetry plan are shown in figure 5 for the four cases. The shear layers from the top and bottom surfaces of the model form a reverse flow zone. Under the effects of the negative pressure on slant and rear end, the upper shear layer exhibits a strong downwash and forms a saddle point, denoted by “x”, with the lower shear layer. The streamwise position of this saddle point can represent the length of the reverse flow zone. For case 1, the downwash becomes more remarkable and the saddle point shifts upstream to the model, which suggests that the reverse flow zone shrinks in case 1. Contrarily, the down-

wash flow becomes weaker and the streamwise extent of reverse flow zone enlarges in cases 2 and 3. For a 2D bluff body, the length of reverse flow zone is in contrast with the aerodynamic drag^[17]. In view of the extended reverse flow zone in cases 2 and 3, the corresponding aerodynamic drag may reduce, which will be discussed in detail later.

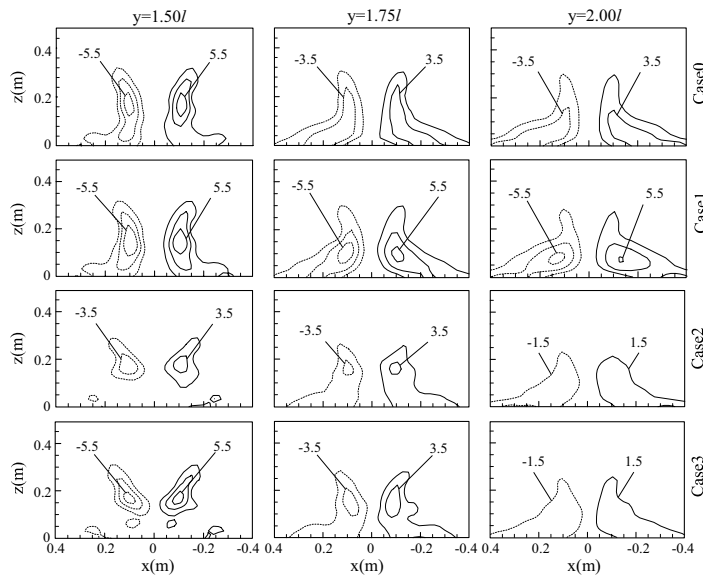


Figure 6. Time-averaged normalized streamwise vorticity ω_y^* for different cases (contour interval=2.0).

Figure 6 presents the time-averaged ω_y^* contours at different downstream positions for the four cases. For case 0, i.e. without control, there is a pair of symmetrical streamwise vortices, which decays monotonously as moving downstream for the three y positions as shown in figure 6. Obviously, this pair of tail vortices is associated with the downwash flow from the slant, as shown in figure 4. Although, the intensity of tail vortices is the same for case 0 and case 1 at $y = 1.5l$, the size of these vortices is slightly larger for the later. For case 1, the vortex intensity does not diminish with increasing y . This observation may be ascribed to the effects of the slot jet along slant surface. The slot jet in case 1 enhances the downwash flow (as shown in figure 5) and subsequently the associated tail vortices. For cases 2 and 3, the application of slot jet noticeably suppresses the tail vortices, especially for the former. For example, at $y = 1.5l$, the maximum of ω_y^* is 3.5, about 64% of that for case 0. The application of slot jet in cases 2 and 3 increases the decaying rate of tail vortices. At $y = 2.0l$, the maximum of ω_y^* is 1.5, about 43% of that for case 0. Since the tail vortices are responsible for a significant part of the aerodynamic drag^[16], the variation of tail vortices as shown in figure 6 suggests that the slot jet has significant effects on the aerodynamic drag.

3.2 Pressure distribution and aerodynamic drag

Pressure coefficients (C_p) on the slant and rear end are presented in figure 7 for the four cases, which is defined as $C_p = 2(p - p_\infty) / \rho U_\infty^2$, where p_∞ is the static pressure of oncoming flow, ρ is the density of air. For case 0, the pressure gradient is remarkable near the top and side edges of the slant surface, suggesting strong flow separation there which forms the tail vortices. For case 1, the slot jet is along the slant, which reinforces the momentum of shear flow and may postpone the flow separation. Consequently, the absolute value of C_p on the slant is slightly smaller for

case 1 than for case 0. However, the absolute value of C_p on rear end for case 1 is larger than that for case 0. That is, the slot jet at the top edge of slant changes not only the pressure distribution on the slant surface, but also alters the whole wake flow. For cases 2 and 3, the extremum of C_p on the slant is -0.25, slightly higher than -0.3 for case 0. On the rear end, although the extremum of C_p for case 2 is the same with case 0, the range of the contour of -0.2 is obviously larger for the former. For case 3, the extremum of C_p is -0.2 one the rear end, slightly smaller than the corresponding value of -0.15 for case 0.

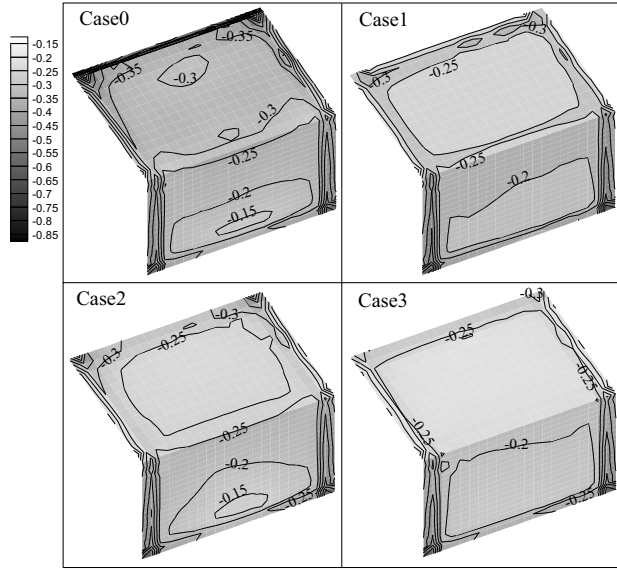


Figure 7. Pressure coefficient on the slant and rear end (contour interval=0.05).

Table 1. Drag coefficients of the model and its different parts.

	Case0	Case1	Case2	Case3
Drag from the front end, C_{D_front}	0.020	0.020	0.020	0.022
Drag from the side walls, $C_{D_sidewall}$	0.043	0.043	0.042	0.042
Drag from the slant, C_{D_slant}	0.103	0.099	0.084	0.091
Drag from the rear end, C_{D_back}	0.164	0.171	0.158	0.163
C_D	0.330	0.333	0.304	0.318
Drag reduction efficiency	---	-0.91%	7.88%	3.64%

To highlight the effects of slot jet on the aerodynamic drag, table 1 lists the overall drag coefficients C_D and its components from different parts of the Ahmed model. For all cases, the drag from the front end of the model is almost unchanged, suggesting that the application of slot jet does not noticeably change the flow near the head. The drag from side walls is largely dominated by the turbulence friction on side walls, which is related to the Reynolds number, wall roughness etc. The present slot jet does not change $C_{D_sidewall}$, similar to that observed for C_{D_front} . Both C_{D_front} and $C_{D_sidewall}$ account for a small portion of total aerodynamic drag. As listed in table 1, most C_D results from the negative pressure portion on the slant and rear end, consistent with that proposed by Drouin *et al.*^[16]. For case 1, although the application of slot jet reduces slightly the drag on slant, C_{D_back} is increased by about 4.3%, which is consistent with the pressure distribution shown in figure 7. The total drag reduction for case 1 is -0.91%, i.e., C_D for case 1 is slightly bigger than that for case 0. Cases 2 and 3 reduce C_D by 7.88% and 3.64%, respectively.

Considering the effects of slot jet on the near wake structure, following observations can be made. Firstly, C_D is inversely correlated with the length of reverse flow zone, i.e. a smaller

length corresponds to a larger C_D . Secondly, C_D is inversely correlated with the strength of tail vortices. A horizontal slot jet, i.e. case 2, suppresses the tail vortices significantly and results in the smallest C_D for present investigated cases. While for case 1, the tail vortices are enhanced and the corresponding C_D is slightly bigger than that without control. Finally, the application of slot jet at the top edge of slant modifies not only the pressure distribution on the slant, but also changes that on the rear end. The total C_D largely comes from the slant and rear end. To evaluate the over all drag reduction efficiency, the combined effects on both slant and rear end should be considered.

4 CONCLUSIONS

A steady slot jet at the top edge of the slant of an Ahmed model is used to control the aerodynamic drag. The slant angle is 25° . Three slot jet directions are investigated using LES technique. For case 1, the slot jet is along the slant surface; for case 2, it is horizontally backwards; and for case 3, it is inclined upwards by 30° relative to the horizontal direction. Based on the simulation results, the following conclusions can be drawn:

(1) There is a pair of symmetrical streamwise tail vortices in the near wake, which is responsible for the negative pressure distribution on the slant and rear end. This negative pressure on slant and rear end accounts for about 80% of the total aerodynamic drag.

(2) The aerodynamic drag is inversely correlated with the length of reverse flow zone and the strength of tail vortices.

(3) The slot jet has strong effects on the near wake and subsequently the aerodynamic drag. For case 1, the length of reverse flow zone shrinks significantly and the tail vortices are enhanced, the corresponding C_D is about 0.91% larger than that without control. While, for cases 2 and 3, the application of slot jet enlarges the reverse flow zone and suppress the tail vortices. The corresponding drag reduction is about 7.88% and 3.64%, respectively.

5 ACKNOWLEDGEMENTS

The authors wish to acknowledge support given to them from the National Natural Science Foundation of China through grants 51108468.

6 REFERENCES

- 1 W. Hucho and G. Sovran, Aerodynamics of road vehicles, Ann. rev. fluid mech., 1993(25) 485-537.
- 2 D. Krentel, R. Muminovic, A. Brunn, W. Nitsche and R. King, Application of active flow control on generic 3D car models. In R. King (Ed.), Active Flow Control II, Springer-Verlag, Berlin Heidelberg, 2010, pp. 223-239.
- 3 S.R. Ahmed, G. Ramm and G. Faltin, Some salient features of the time-averaged ground vehicle wake, SAE Society of Automotive Eng., 1984, 1(840300): 1-31.
- 4 S. Krajnovic and L. Davidson, Flow around a simplified car, Part 1: Large eddy simulation, J. Fluids Eng., 2005(127) 907-918.
- 5 S. Krajnovic and L. Davidson, Flow around a simplified car, Part 1: Understanding the flow, J. Fluids Eng., 2005(127) 919-928.
- 6 P. Gillieron and A. Kourta, Aerodynamic drag reduction by vertical splitter plates, Exp. Fluids, 2010(48) 1-16.
- 7 P. Joseph, X. Amandolese and J.L. Aider, Drag reduction on the 25° slant angle Ahmed reference body using pulsed jets, Exp. Fluids, 2011 DOI 10.1007/s00348-011-1245-5.
- 8 R. Verzicco, M. Fatica, G. Iaccarino, P. Moin and B. Khalighi, Large Eddy simulation of a road vehicle with drag-reduction devices, AIAA J. 2002(40) 2447-2455.
- 9 J.F. Beaudoin and J.L. Aider, Drag and lift reduction of a 3D bluff body using flaps, Exp. Fluids, 2008(44) 491-501.
- 10 G. Fourrie, L. Keirsbulck, L. Labraga and P. Gillieron, Bluff-body drag reduction using a deflector, Exp. Fluids, 2011(50) 385-395.

- 11 M. Roumeas, P. Gillieron and A. Kourta, Separated flows around the rear window of a simplified car geometry, *J Fluid Eng.*, 2008(6) 1229-1139.
- 12 S. Krajnovic, J. Östh and B. Basara, LES of active flow control around an Ahmed body with active flow control, In: Conference on modeling fluid flow (CMFF'09), 2009, Budapest.
- 13 J. Smagorinsky, General circulation experiments with the primitive equations, *Mon. Weather Rev.*, 1963(91) 99-165.
- 14 A. Brunn, E. Wassen, D. Sperber, W. Nitsche and F. Thiele, Active drag control for a generic car model, In R. King (Ed.), *Active Flow Control*, Springer-Verlag, Berlin Heidelberg, 2007, 247-259.
- 15 G. Vio, S. Watkins, P. Mousley, J. Watmuff and S. Prasad, Flow structures in the near-wake of the Ahmed model, *J. Fluids. Struc.*, 2005(20) 673-695.
- 16 V. Drouin, A. Giovannini and P. Gillieron, Topology and characterization of the vertical near-wake flow over a simplified car model. In *Proceedings of the Bluff body wakes and vortex induced vibrations (BBVIV 3) conference*, 2002, Port-Douglas, Australia.
- 17 S. Balachandar, R. Mittal and F.M. Najjar, Properties of the mean recirculation region in wakes of two-dimensional bluff bodies, *J. Fluid Mech.*, 1997(351), 167–199.