Lecture Notes in Applied and Computational Mechanics

Volume 41

Lecture Notes in Applied and Computational Mechanics

Edited by F. Pfeiffer and P. Wriggers

Further volumes of this series found on our homepage: springer.com

Vol. 41: Browand F., Ross J., McCallen R. (Eds.)
Aerodynamics of Heavy Vehicles II: Trucks, Buses, and Trains
486 p. 2009 [978-3-540-85069-4]

Vol. 40: Pfeiffer F.
Mechanical System Dynamics
578 p. 2008 [978-3-540-79435-6]

Vol. 39: Lucchesi, M.; Padovani, C.; Pasquinelli, G.; Zani, N.
Masonry Constructions: Mechanical Models and Numerical Applications
176 p. 2008 [978-3-540-79110-2]

Vol. 38: Marynowski, K.
Dynamics of the Axially Moving Orthotropic Web
140 p. 2008 [978-3-540-78988-8]

Vol. 37: Chaudhary, H.; Saha, S.K.
Dynamics and Balancing of Multibody Systems
200 p. 2008 [978-3-540-78178-3]

Vol. 36: Leine, R.I.; van de Wouw, N.
Stability and Convergence of Mechanical Systems with Unilateral Constraints
250 p. 2008 [978-3-540-76974-3]

Vol. 35: Acary, V.; Brogliato, B.
Numerical Methods for Nonsmooth Dynamical Systems: Applications in Mechanics and Electronics
545 p. 2008 [978-3-540-75391-9]

Vol. 34: Flores, P.; Ambrósio, J.; Pimenta Claro, J.C.; Lankarani Hamid M.
Kinematics and Dynamics of Multibody Systems with Imperfect Joints: Models and Case Studies
186 p. 2008 [978-3-540-74359-0]

Vol. 33: Niesłony, A.; Macha, E.
Spectral Method in Multiaxial Random Fatigue
146 p. 2007 [978-3-540-73822-0]

Vol. 32: Bardzokas, D.I.; Filshtinsky, M.L.; Filshtinsky, L.A. (Eds.)
Mathematical Methods in Electro-Magneto-Elasticity
530 p. 2007 [978-3-540-71030-1]

Vol. 31: Lehmann, L. (Ed.)
Wave Propagation in Infinite Domains
186 p. 2007 [978-3-540-71108-7]

Vol. 30: Stupkiewicz, S. (Ed.)
Micromechanics of Contact and Interphase Layers
206 p. 2006 [978-3-540-49716-5]

Vol. 29: Schanz, M.; Steinbach, O. (Eds.)
Boundary Element Analysis
571 p. 2006 [978-3-540-47465-4]

Vol. 28: Helmig, R.; Mielke, A.; Wohlmuth, B.I. (Eds.)
Multifield Problems in Solid and Fluid Mechanics
571 p. 2006 [978-3-540-34959-4]

Vol. 27: Wriggers P., Nackenhorst U. (Eds.)
Analysis and Simulation of Contact Problems
395 p. 2006 [978-3-540-31760-9]

Vol. 26: Nowacki, J.P.
Static and Dynamic Coupled Fields in Bodies with Piezoeffects or Polarization Gradient
209 p. 2006 [978-3-540-31668-8]

Vol. 25: Chen C.-N.
Discrete Element Analysis Methods of Generic Differential Quadratures
282 p. 2006 [978-3-540-28947-0]

Vol. 24: Schenk, C.A., Schuëller. G
Uncertainty Assessment of Large Finite Element Systems
165 p. 2006 [978-3-540-25343-3]

Vol. 23: Frémond M., Maceri F. (Eds.)
Mechanical Modelling and Computational Issues in Civil Engineering
400 p. 2005 [978-3-540-25567-3]

Vol. 22: Chang C.H.
Mechanics of Elastic Structures with Inclined Members: Analysis of Vibration, Buckling and Bending of X-Braced Frames and Conical Shells
190 p. 2004 [978-3-540-24384-7]

Vol. 21: Hinkelmann R.
Efficient Numerical Methods and Information-Processing Techniques for Modeling Hydro- and Environmental Systems
305 p. 2005 [978-3-540-24146-1]

Vol. 20: Zohdi T.I., Wriggers P.
Introduction to Computational Micromechanics
196 p. 2005 [978-3-540-22820-2]

Vol. 19: McCallen R., Browand F., Ross J. (Eds.)
The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains
567 p. 2004 [978-3-540-22088-6]

Vol. 18: Leine, R.I., Nijmeijer, H.
Dynamics and Bifurcations of Non-Smooth Mechanical Systems
236 p. 2004 [978-3-540-21987-3]

Vol. 17: Hurtado, J.E.
Structural Reliability: Statistical Learning Perspectives
257 p. 2004 [978-3-540-21963-7]

Vol. 16: Kienzler R., Altenbach H., Ott I. (Eds.)
Theories of Plates and Shells: Critical Review and New Applications
238 p. 2004 [978-3-540-20997-3]

Vol. 15: Dyszlewicz, J.
Micropolar Theory of Elasticity
356 p. 2004 [978-3-540-41835-1]

The Aerodynamics of Heavy Vehicles II: Trucks, Buses, and Trains

Fred Browand
Rose McCallen
James Ross (Eds.)

With 308 Figures and 52 Tables

Springer

Fred Browand, Ph.D.
Department of Aerospace
and Mechanical Engineering
University of Southern California
Los Angeles, CA 90089-1191
USA

Rose McCallen, Ph.D.
Center for Advanced
Fluid Dynamics
Applications (CAFDA)
Lawrence Livermore
National Laboratory
P.O.Box 808, L-098
Livermore CA 94551
USA

James Ross, Ph.D.
NASA Ames Research Center
Experimental Physics Branch
MS 260-1
Moffett Field CA 94035
USA

ISBN: 978-3-540-85069-4 e-ISBN: 978-3-540-85070-0

Lecture Notes in Applied and Computational Mechanics ISSN 1613-7736
e-ISSN 1860-0816

Library of Congress Control Number: 2008934051

Cover design: WMXDesign GmbH, Heidelberg

Printed on acid-free paper

9 8 7 6 5 4 3 2 1 0

springer.com

Engineering Conferences International

Engineering Conferences International (ECI) is a not-for-profit global engineering conferences program, originally established in 1962, that provides opportunities for the exploration of problems and issues of concern to engineers and scientists from many disciplines. The organization sponsors leading edge, interdisciplinary, international scientific/engineering conferences.

Executive Director:
Barbara K. Hickernell
Engineering Conferences International
32 Broadway, Suite 314
New York, NY 10004
USA
Tel: 1-212-514-6760
Email: info@engconfintl.org
www.engconfintl.org

Introduction

It is our pleasure to present these proceedings for "The Aerodynamics of Heavy Vehicles II: Trucks, Buses and Trains" International Conference held in Lake Tahoe, California, August 26-31, 2007 by Engineering Conferences International (ECI). Brought together were the world's leading scientists and engineers from industry, universities, and research laboratories, including truck and high-speed train manufacturers and operators. All were gathered to discuss computer simulation and experimental techniques to be applied for the design of the more efficient trucks, buses and high-speed trains required in future years.

This was the second conference in the series. The focus of the first conference in 2002 was the interplay between computations and experiment in minimizing aerodynamic drag. The present proceedings, from the 2007 conference, address the development and application of advanced aerodynamic simulation and experimental methods for state-of-the-art analysis and design, as well as the development of new ideas and trends holding promise for the coming 10-year time span. Also included, are studies of heavy vehicle aerodynamic tractor and trailer add-on devices, studies of schemes to delay undesirable flow separation, and studies of underhood thermal management.

We would like to thank the ECI organizers for their efficient organization of the meeting. In addition, we would like to express our appreciation to all session chairs, the scientific advisory committee, authors, and reviewers for their many hours of dedicated effort that contributed to a successful conference, and that are manifest in this proceeding. We also gratefully acknowledge the financial support received from ECI, the United State's Truck Manufacturers Association, International Truck and Engine Corporation, Lawrence Livermore National Laboratory, and CD Adapco.

Conference Co-Chairs:

Professor Fred Browand, Ph.D.
Aerospace & Mechanical Engineering Department
University of Southern California
Los Angeles, CA

Rose McCallen, Ph.D.
Lawrence Livermore National Laboratory
Livermore, CA

Jim Ross, Ph.D.
NASA Ames Research Center
Moffett Field, CA

Contents

Keynote Papers

Flow Field Characteristics

Separation Control for Drag Reduction

Train Aerodynamics

Poster Session

CFD, Numerical Methods and Application

Vehicle and Tire Spray and Vehicle Interaction

Drag Reduction

Keynote Papers

Bluff Body Flow Research with Application to Road Vehicles

Peter Bearman

Department of Aeronautics, Imperial College, London, SW7 2AZ, UK
p.bearman@imperial.ac.uk

Abstract This paper explores a number of aspects of bluff body research that may help in the understanding and advancement of the aerodynamics of heavy vehicles. The relationship between the lift and drag of bodies moving close to the ground is discussed and the unsteady trailing vortex structure of a vehicle is illustrated. The importance of the ground boundary condition and the flow structure around wheels rotating in contact with the ground are also described. Methods for the reduction of forebody and base drag are discussed and the possibilities for using flow control techniques to modify free shear layer development as a means of reducing drag are addressed. Finally the role of the natural wind, particular atmospheric turbulence, in affecting vehicle flows is examined.

Introduction

Road vehicles are subject to design constraints which severely limit aerodynamic efficiency and practical vehicles operate with regions of separated flow. While much of our understanding of road vehicle aerodynamics has developed through the application of techniques used in the aerospace industry, vehicles fall into a category of bodies that are aerodynamically bluff. For streamlined bodies considerable advances are being made in using flow control methods to reduce skin friction drag but with bluff bodies the emphasis needs to be on minimising pressure drag. This means keeping flow attached over as much of the body surface as possible and raising the pressure in the large separated region at the rear of a vehicle. For economic, environmental and political reasons, drag reduction is a prime goal but it should be kept in mind that aerodynamics affects other important aspects of vehicle operations including: cooling, handling and noise. However, in this paper the main emphasis is placed on the processes of drag generation and drag reduction of road vehicles.

Aerodynamics is a key technology in motor racing, particularly for the open-wheeled vehicles used, for example, in Formula 1. Here significant advances have been made in wind tunnel test techniques and in the appreciation of the importance of the correct simulation of the ground boundary condition. This has led to improved understanding of lift generating mechanisms and the complex problem of wheel flow, which in one form or another affects all road vehicles. A further consideration is that vehicles are exposed to the natural wind and as a result can experience yawed flow and aerodynamic loading influenced by turbulence. Here there is an obvious connection with the study of wind loading on buildings and structures. Hence in addressing the aerodynamics of heavy road vehicle, ideas will be drawn from aerospace, wind engineering, motor racing, as well as from basic research on bluff bodies. In summary, some of the important aerodynamic considerations for road vehicles are:

- Large areas of separated flow leading to high drag
- Close proximity to the ground which influences both lift and drag
- The presence of wheels
- The effects of the natural wind introducing yawed flow and turbulence
- Interference due to the close proximity of other vehicles

The Relation between Lift and Drag

For an aircraft, lift is accompanied by the generation of trailing vortices and these vortices contribute to induced or lift-related drag. Total drag is then the summation of profile drag and induced drag. However, for road vehicles the relationship between lift and drag is more complex because even in an ideal, or inviscid, flow lift can be generated on a body moving close to a boundary. This manifests itself as an attractive force between the body and its imaginary image in the boundary. For example, a sphere moving near the ground through an ideal fluid generates a force towards the ground and the associated lift coefficient can rise to nearly -0.4 when the gap between the sphere surface and the ground is very small. This force is generated without the shedding of trailing vortices and without drag. For a road vehicle the lift will be a combination of this component and the more familiar one experienced by lifting bodies in a real fluid. It is the latter component that is related to trailing vortices and drag generation.

Most vehicles generate lift but the associated trailing vortices are unsteady and rapidly dissipate since they are influenced by the large separated region at the rear of a vehicle. Figure 1 shows two velocity maps measured in the near wake of a passenger car model using particle image velocimetry and reported by Wang et al (1996). They were measured a short time apart and it is clear that there are substantial differences in the two plots. However, if a small number of these maps are averaged then the time-average picture that appears is quite different. Figure 2 shows the result of averaging just 10 plots and now we see the emergence of a pair of trailing vortices but it is clear that the vortex structure in the instantaneous flow is very different to the time-average one. Such observations give strength to the argument that LES methods are likely to be more appropriate for computing vehicles flows than using the RANS equations coupled with a turbulence model.

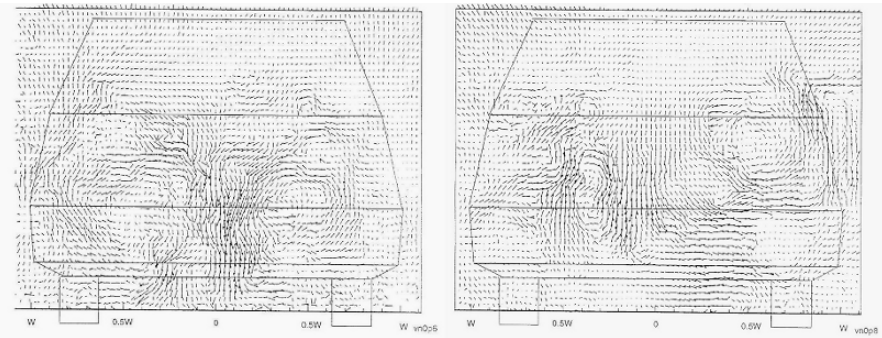

Fig. 1 Instantaneous cross-flow velocity maps measured in the near wake of a passenger car model at a vertical plane 20% of the car length from the rear bumper

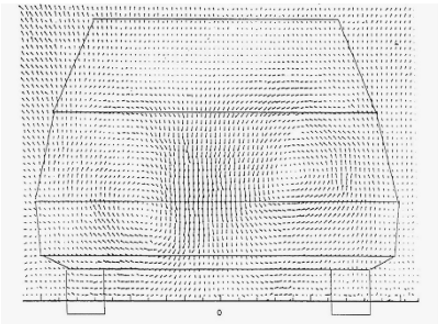

Fig. 2 The velocity field obtained from the average of only 10 instantaneous velocity maps

The Ground Boundary Condition and Wheels

Wind tunnel testing using a rolling road has become standard practice for many racing car applications where an important feature is the low ground clearance. As well as generating the correct ground boundary condition, a further advantage of using a rolling road is that wheels and their effect on aerodynamic performance can be correctly simulated. Although at first sight there may not appear to be too many similarities between a Formula 1 racing car and a tractor-trailer vehicle, when ground clearance is expressed as a fraction of vehicle length the values for the two are not so different. Also for both vehicles the wheel flows are important, particularly for a trailer where wheels are often very exposed. If vehicle flow is considered in a wind tunnel frame of reference where the ground is correctly simulated, an obvious observation is that the relative flow speeds beneath and over a vehicle are similar. This has an important implication for drag reduction since there are many potential drag generating components under a vehicle and designers need to pay as much attention to the design of the underbody as they do to the more visible parts of a vehicle.

Obtaining a full understanding of the flow around a rotating road wheel presents a challenging problem. Working in a wind tunnel frame of reference, the flow travelling along the ground meets flow driven by the no slip condition at the tyre surface, which is a thin layer of flow rotating with the wheel, in the region of contact between the wheel and the ground. The result of these flows coming together is the sideways jetting flow that can be seen in the instantaneous PIV velocity field plotted in Fig. 3. Here the measurement plane is very close to the ground and parallel to it and the free stream flow is from left to right. The jet and resulting large region of locally disturbed flow are clearly visible in this plot from a study of a racing car wheel by Pegrum (2006). The investigation of this problem is made even more complex because contact with the road occurs over a patch of tyre and the contact area may change in response to varying driving conditions.

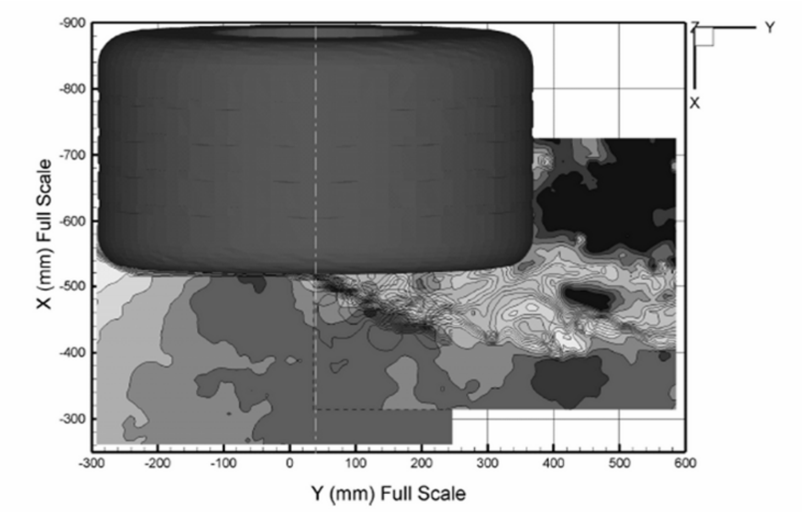

Fig. 3 An instantaneous velocity field measured in a horizontal plane just above the contact patch of a rotating wheel (Pegrum (2006))

As discussed by Fackrell and Harvey (1975) in their pioneering work on wheel flow, separation from the circumferential tyre surface is affected strongly by wheel rotation. Again working in a stationary frame of reference, at the top of the wheel the flow at the surface is moving in opposition to the free stream flow. For a rotating wheel flow separation is brought forward compared to that for a non-rotating one. Reynolds number effects on the flow about a rotating wheel are far less than those for the classic problem of a fixed circular cylinder where a large drop in drag coefficient occurs in the critical Reynolds number regime. Zdravkovich et al (1998) have termed small aspect ratio circular cylinders “coin-like” bodies where separation from the front edges is as important, or perhaps more important, than separation from the curved boundary. Wheel flows fall into a similar category and they give rise to the shedding of a vortex system that is quite different to that for a two-dimensional cylinder. Figure 4 shows a plot of vorticity passing through a vertical plane placed 20% of the wheel diameter behind the back of a wheel. This data, obtained by Pegrum (2006) using PIV, shows two pairs of counter-rotating vortices, one pair emanating from the upper edges of the tyre and

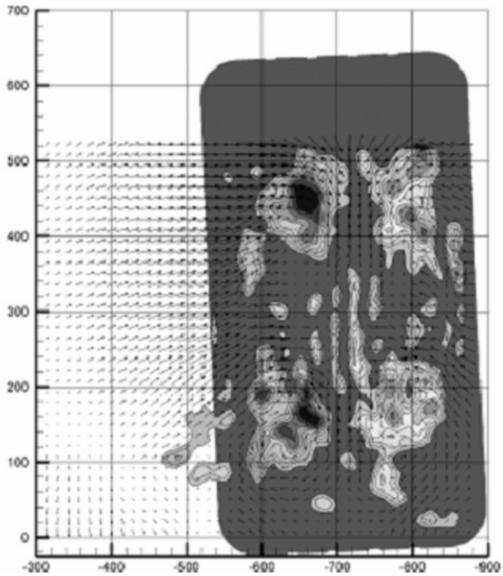

Fig. 4 Vorticity field showing the wake structure at 20% of a diameter behind a wheel (Pegrum (2006))

the other forming lower down in the near wake. The self-induced velocity on the upper pair drives the vortices down and they merge with the lower pair. Hence further downstream only one pair of vortices can be found. Figure 5, due to Saddington et al(2007), shows how the vortex configuration changes with distance behind an isolated wheel. It should be noted that wheel wake flows are highly unsteady and the sketches in Fig. 5 present a time-average picture.

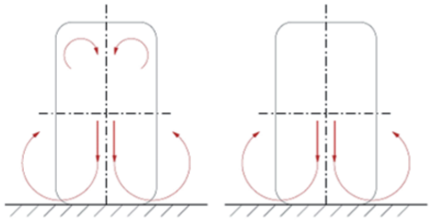

Fig. 5 Vortex Structure behind a Rotating Wheel, left 0.5 diameters downstream and right 2 diameters downstream (Saddington et al (2007))

Drag Reduction of Heavy Vehicles

As mentioned earlier, possibilities for drag reduction of heavy vehicles are limited by the requirement that the basic function of the vehicle should not be compromised. This puts a serious constraint on what might be done to reduce the base drag of a trailer, for example. The aerodynamic drag of a heavy vehicle results primarily from differences in pressure acting on forward and rearward facing surfaces and for the simplified vehicle shape shown in Fig. 6 the drag can be conveniently split into forebody and base drag components. The techniques for reducing these two components differ. To reduce the forebody drag, regions of flow separation need to be minimised. For the basic body shown in Fig. 6 this might be done by increasing corner radii and by enhancing the mixing in the separated shear layer to encourage earlier reattachment. The more difficult problem is to reduce base drag because the practical constraints are much more severe here. To increase the pressure in the base region some form of mild boat-tailing might be used to raise the pressure ahead of separation. Alternatively, since the base pressure is related to the entrainment of fluid into the surrounding separated shear layers, the base pressure would increase if the mixing in the separated shear layers could be reduced. Modern developments in flow control technology are providing interesting possibilities for achieving this.

Techniques that have been used, and also those that might be used in the future, to reduce the drag of heavy road vehicles include:

- Shielding, incorporating the strategy of driving in close platoons
- The long established method of fitting cab-top deflectors to reduce forebody drag
- Surface treatments to delay separation, possibly including the use of dimples
- Base cavities and boat-tailing to reduce base drag
- Active and passive methods applied to the control of separation
- Active and passive methods applied to the control of separated shear layer development

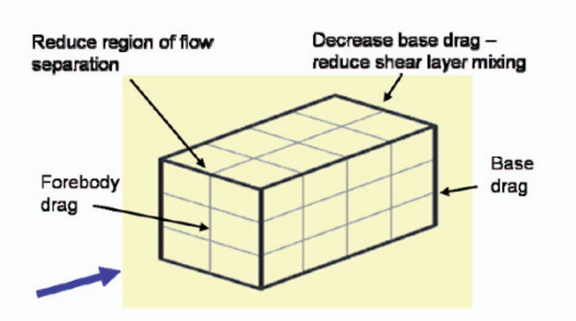

Fig. 6 A highly simplified heavy road vehicle

Considerable success can be achieved in reducing drag by attention to a large number of small details on a vehicle. This is sometimes referred to as aerodynamic tuning but it can only be taken so far and a significant region of separated flow is always likely to remain at the rear of a vehicle. For a bluff body with fixed separation, say from sharp edges, the higher the base suction then the smaller the time-average recirculation region in the near wake. Hence, to reduce drag the recirculation region must be made larger, not smaller. This can be achieved by reducing stresses in the free layers so as to reduce entrainment from the near wake and hence raise the base pressure. The challenging problem this poses is how can the structure of separated shear layers be controlled in such a way as to reduce stresses?

Chun and Sung(1996) have carried out an experimental study of flow over a backward-facing step where the separating shear layer has been excited by an oscillating jet. The experimental set up is shown in Fig.7 and it can be seen from the detail that the massless jet issues from a small slit at the corner of the step. Some of the results from this experiment are reproduced in Fig. 8 where the mean flow reattachment distance behind the step, X_r, non-dimensionalised by the reattachment distance with no blowing, is plotted against non-dimensional forcing frequency. Here the frequency of forcing is non-dimensionalised by using the step height and the approaching flow velocity to form the parameter St_H. The results for three jet velocity amplitudes are plotted and they all show similar features. As frequency is increased the reattachment length shortens, a minimum is reached and for high values of frequency the reattachment length extends beyond that for no jet. An interpretation of these results is that the Kelvin-Helmholtz type instability in the shear layer is enhanced at low values of the frequency parameter leading to increased entrainment, higher base suction and a shorter reattachment length. From the viewpoint of drag reduction, the significant result is the increase in reattachment length for high forcing frequencies where presumably the high frequency massless jet interferes in a beneficial way with the shear layer instability mechanism leading to reduced stresses. This will result in a lower base drag on the step.

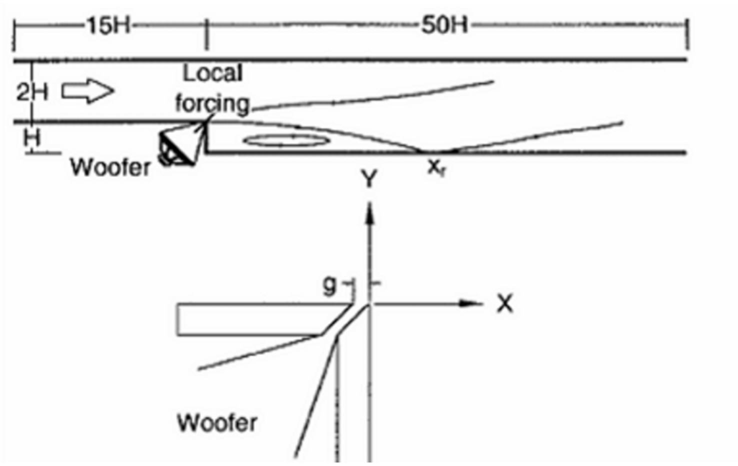

Fig. 7 Control of flow over a backward-facing step (Chun and Sung (1996))

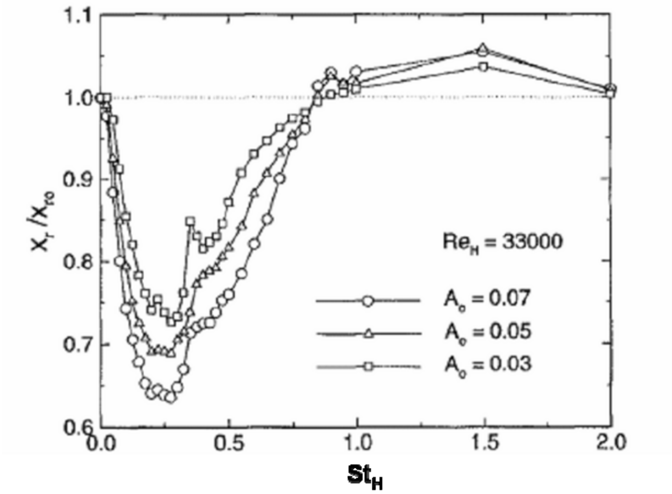

Fig. 8 Reattachment length behind a backward-facing step versus non-dimensional forcing frequency (Chun and Sung (1996))

A further example of the effectiveness of a massless jet can be found in the numerical simulations of Dandois et al (2007). Figure 9 shows examples of their direct numerical simulations of flow over a rounded two-dimensional, backward-facing ramp. The upper plot shows the mean flow without control, the middle one is with low frequency forcing and the effect of high frequency forcing is shown in the lower plot. Again enhanced entrainment occurs for low frequency forcing and some suppression takes place for high frequency forcing leading to a substantial increase in reattachment length. The application of such techniques to reduce the drag of road vehicles appears possible but further research is required to determine the overall efficiency of such a control method, its optimisation and practicality. Nevertheless, it presents an interesting new direction.

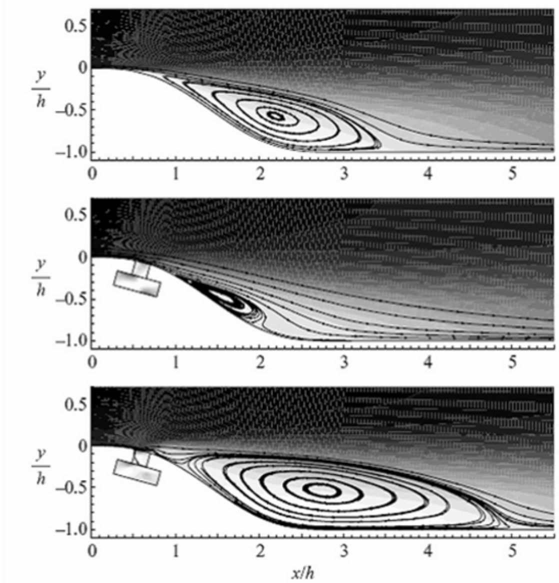

Fig. 9 Numerical simulations of flow control over a two-dimensional ramp using a massless jet: upper no control, middle low frequency forcing, lower high frequency forcing (Danois et al (2007))

The Influence of the Natural Wind on Road Vehicle Flows

Road vehicles are subjected to strong flow disturbances generated by other vehicles and by the natural wind. The influence of the wind can be summarised as follows:

- Changes occur in the magnitude and direction of the relative approaching flow
- Small angles of yaw can lead to increases in aerodynamic drag and the generation of side forces that may give rise to handling problems
- Turbulence produced by the wind generates unsteady aerodynamic forces
- The presence of turbulence may change mean force coefficients

Bearman and Morel (1983) reviewed the effects of free stream turbulence on the flow around bluff bodies and identified two mechanisms by which turbulence can influence the mean flow about a body. These are shown in diagrammatic form in Fig. 10 for an attached boundary layer and a free shear layer. For vehicles, turbulence can affect separation, reattachment and the pressure in a separated region.

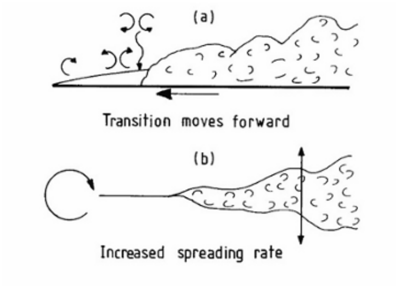

Fig. 10 Two mechanisms by which free stream turbulence affects flows (Bearman and Morel (1983))

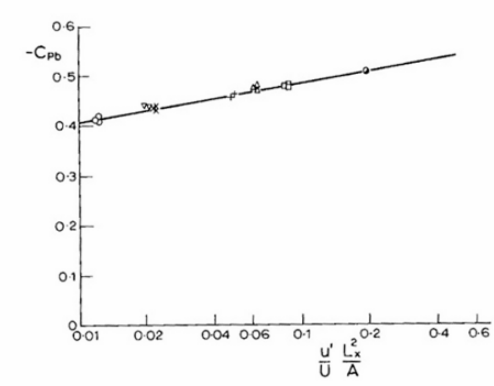

Fig. 11 Base pressure coefficients acting on square and circular thin plates set normal to turbulent flow

Figure 11 shows the effect of turbulence on the base pressure acting on thin plates mounted normal to a flow. Base pressure coefficient is seen plotted against a turbulence parameter involving the product of the free stream turbulence intensity and the ratio of turbulence scale length squared to plate area. This parameter is discussed by Bearman (1971) and has been devised on the assumption that the main effect of turbulence for the plates is to enhance mixing in their near wakes. While buildings may be exposed to turbulence levels of 20% or more, vehicles that are moving in the presence of wind see substantially lower relative intensities under conditions of high vehicle speeds. A practical range for the turbulence parameter used in Fig.11 may be up to 0.06 but as can be seen from the results this can still lead to a significant increase in base suction.

The interaction between a turbulent approaching flow and a road vehicle is complex and gives rise to a number of different effects that are too varied to go into further detail here. However, let it suffice to say that turbulence gives rise to uncertainties when comparing on road measurements of aerodynamic quantities with measurements made in a conventional smooth flow wind tunnel.

References

Bearman P (1971) Investigation of forces on flat plates normal to a turbulent flow. J Fluid Mech 46:177-198

Bearman P, Morel T (1983) Effect of free stream turbulence on the flow around bluff bodies. Prog. Aerospace Sci 20:97-123

Chun K, Sung H (1996) Control of turbulent separated flow over a backward-facing step by local forcing. Exp Fluids 21:417-426

Danois J, Garnier E, Sagaut P (2007) Numerical simulation of active separation control by a synthetic jet. J Fluid Mech 574:25-58

Fackrell J, Harvey J (1974) The aerodynamics of an isolated road wheel. Proc 2nd AIAA Symp. of Aerodynamics of Sports and Competition Automobiles, 16:119-125

Pegrum J (2006) Experimental study of the vortex system generated by a Formula 1 front wing. PhD thesis, University of London

Saddington A, Knowles R, Knowles K (2007) Laser Doppler anemometry measurements in the near-wake of an isolated Formula One wheel. Exp Fluids 42:671-681

Wang Q, Bearman P, Harvey J (1996) A study of the instantaneous flow structure behind a car by particle image velocimetry. International Seminar on Optical Methods and Data Processing in Heat and Fluid Flow, 1996:179-188, IMechE, London

Zdravkovich M, Flaherty A, Pahle M, Skelhorne I (1998) Some aerodynamic aspects of coin-like cylinders. J Fluid Mech 360:73-84

Climate Change and the Energy Economy

David Schimel

The National Center for Atmospheric Research (NCAR), USA
schimel@ucar.edu

Abstract There is a robust scientific consensus that human-induced climate change is occurring. The recently released Fourth Assessment Report of the IPCC states with "very high confidence," that human activity has caused the global climate to warm. Many well-documented observations show that fossil fuel burning, deforestation, and other industrial processes are rapidly increasing the atmospheric concentrations of CO_2 and other greenhouse gases. An increasing body of observations and modeling results shows that these changes in atmospheric composition are changing the global climate and beginning to affect terrestrial and marine ecosystems. In this talk, I'll review observed and projected changes to the US climate, discuss the sectoral contributions to US greenhouse gas emissions and speculate about the future of the US carbon economy.

Flow Field Characteristics

Large Eddy Simulations and Acoustic Predictions in Automotive Applications

G. Iaccarino[1], F. Ham[1], Y. Khalighi[1], D. Bodony[1], P. Moin[1], B. Khalighi[2]

[1] Stanford University, USA
[2] General Motors R&D Center, USA
jops@stanford.edu (G. Iaccarino)

Abstract The emergence of the computational tools capable of accurately predicting the flow properties around complex geometries has opened new opportunities in the design of automotive components and their integrations. In particular, one may now be able to predict the acoustic noise produced by the unsteady fluid motion induced by the individual parts and quantitatively evaluate their loudness and spectral (i.e. frequency content) without relying on semi-empirical models. To demonstrate this capability the flow noise produced by two configurations typical of automotive applications – namely the outside rear view mirrors and the rain gutters – are examined quantitatively. Relevant details regarding the prediction methods as well as a selection of results are discussed.

Validation of the Immersed Boundary CFD Approach for Complex Aerodynamic Flows

B. Khalighi[1], S. Jindal[1], J.P. Johnson[1], K.H. Chen[1], G. Iaccarino[2]

[1] General Motors, USA
[2] Stanford University, USA
bahram.khalighi@gm.com

Abstract Standard CFD methods require a mesh that fits the boundaries of the computational domain. For a complex geometry the generation of such a grid is time-consuming and often requires modifications to the model geometry. This paper evaluates the Immersed Boundary (IB) approach which does not require a boundary-conforming mesh and thus would speed up the process of the grid generation. In the IB approach the CAD surfaces (in Stereo Lithography –STL- format) are used directly and this eliminates the surface meshing phase and also mitigates the process of the CAD cleanup. A volume mesh, consisting of regular, locally refined, hexahedrals is generated in the computational domain, including inside the body. The cells are then classified as fluid, solid and interface cells using a simple ray-tracing scheme. Interface cells, correspond to regions that are partially fluid and are intersected by the boundary surfaces. In those cells, the Navier-Stokes equations are not solved, and the fluxes are computed using geometrical reconstructions. The solid cells are discarded, whereas in the fluid cells no modifications are necessary. The present IB method consists of two main components: 1) TOMMIE which is a fast and robust mesh generation tool which requires minimum user intervention and, 2) a library of User Defined Functions for the FLUENT CFD code to compute the fluxes in the interface cells. This study evaluates the IB approach, starting from simple geometries (flat plate at 90 degrees, backward facing step) to more complex external aerodynamics of full-scale fully-dressed production vehicles. The vehicles considered in this investigation are a sedan (1997 Grand-Prix) and an SUV (2006 Tahoe). IB results for the flat plate and the backward-step are in very good agreement with measurements. Results for the Grand-Prix and Tahoe are compared to experiments (performed at GM wind tunnel) and typical body-fitted calculations performed using Fluent in terms of surface pressures and drag coefficients. The IB simulations predicted the drag coefficient for the Grand-Prix and the Tahoe within 5% of the body-fitted calculations and are closer to the wind-tunnel measurements.

Introduction

Numerical simulations allow the analysis of complex phenomena without resorting to expensive prototypes and difficult experimental measurements. This trend, to bring CFD to bear on vehicle aerodynamics design issues, is appropriate and timely in view of the increasing competitive and regulative pressures being faced by the automotive industry [1].

The industrially relevant geometries are usually defined in the CAD environment and must be *cleaned* (small details are usually eliminated and overlapping surface patches are trimmed- before it can be used for computational analysis). A smooth water-tight surface mesh is then generated to serve as a boundary condition for the volume mesh. There are mainly two types of approaches in volume meshing, structured and unstructured meshing. In structured mesh, the governing equations are transformed into the curvilinear coordinate system aligned with the surface. It is trivial for simple shapes, however, becomes extremely inefficient and time consuming for complex geometries. In the unstructured approach, there is no transformation involved for governing equations. The integral form of governing equations is discretized and either a finite-volume or finite-element scheme is used. The information regarding the grid is directly incorporated into the discretization. Unstructured grids are in general successful for complex geometries. However, the quality of these grids deteriorates with complex shapes. In addition, there is large computational overhead owing to a large number of operations per node and low accuracy in combination with the low-order dissipative spatial discretization.

The Immersed Boundary (IB) method [2] is an alternative procedure that can handle the geometric complexity, and at the same time retains the accuracy and high efficiency of the simulations performed on regular grids. This represents a significant advance in the application of CFD to realistic flows. It uses Cartesian-like meshes in a simple, fictitious computational domain obtained by eliminating the object of interest (i.e. the road vehicles in the present application).

The main objective of this study is to evaluate the IB method to assess the accuracy, robustness, and the speed for routine aerodynamics simulations in the automotive industry. The IB flow simulations are presented for two simple geometries as well as for two production vehicles. The vehicles considered in this investigation are a sedan (1997 Pontiac Grand-Prix) and an SUV (2006 Chevy Tahoe).

The IB Method

The IB method is an alternative procedure in which a non-body conformal grid is used. The method “immersed boundary” was first developed by [3] to simulate cardiac mechanics and the associated blood flows. In the present approach, the boundary surfaces are still present but the volume mesh is generated independently. Because the volume grid does not conform to the solid boundary, enforcing the physical boundary conditions requires a modification of the equations in the vicinity of boundary. Consider the Navier-Stokes equations below,

$$\pounds(\underline{U}) = 0 \text{ in } \Omega_f \text{ (volume)} \tag{1}$$

$$\text{with} \qquad \underline{U} = \underline{U}_{\Gamma} \text{ on } \Gamma_b \text{ (surface)} \tag{2}$$

Where $\underline{U} = (u,p)$ and $\pounds$ are the operators representing the Navier-Stokes equations. u and p refer to the velocity components and the pressure, respectively. Conventional methods proceed by developing a discretization of equation (1) on a body-conformal grid where the boundary condition (equation 2) on the immersed surface Γ_b is enforced directly. For IB, a forcing function is used in the governing equations that reproduce the effect of the boundary.

$$\pounds(\underline{U}) = f_b \quad \text{(volume + surface)} \tag{3}$$

The above system of equations is solved in the entire fluid domain. The forcing function f_b can be defined before and after the discretization step which leads to a dichotomy among different IB methods. A detailed discussion is provided in [4]. In the present approach, the forcing is defined after the discretization step and is applied in all the cells cut by the immersed surfaces (interface cells). The forcing is computed such that the desired boundary value is recovered at the location of the immersed boundary. In finite volume methods, source terms can be interpreted as flux imbalances and, in the current implementation of the IB method, the forcing is actually replaced by a modified flux applied at the faces connecting fluid and interface cells. No modification (f_b =0) is applied at the fluid cells.

In the IB approach, the ramifications of the boundary treatment on the accuracy and the conservation properties of the numerical scheme are not trivial. In the body-fitted method the conformal grid aligns the gridlines and the body surface and allows better control of the grid resolution in the vicinity of the body. However the same is not true for the IB methodology and this results in an increase in the grid size with the Reynolds number for the IB. As described by [4], the grid size ratio would scale to $(\mathrm{Re})^{1.5}$ for a 3D body.

A schematic diagram depicting the CAD to the CFD solution in the IB framework is shown in Fig. 1. The IB approach in this study consists of two main parts; 1) TOMMIE, a mesh generation tool, and 2) IBLIB, a set of User Defined Functions (UDFs) for the FLUENT CFD code which are described briefly in the following sections.