

Fred Browand
Rose McCallen
James Ross (Eds.)

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



Lecture Notes in Applied and Computational Mechanics

Volume 41

Series Editors

Prof. Dr.-Ing. Friedrich Pfeiffer
Prof. Dr.-Ing. Peter Wriggers

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:** Nieslony, A.; Macha, E.
Spectral Method in Multiaxial Random Fatigue
146 p. 2007 [978-3-540-73822-0]
- Vol. 32:** Bardzokas, D.I.; Filstinsky, M.L.; Filstinsky, 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



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

© Springer-Verlag Berlin Heidelberg 2009

This work is subject to copyright. All rights are reserved, whether the whole or part of the material is concerned, specifically the rights of translation, reprinting, reuse of illustrations, recitation, broadcasting, reproduction on microfilm or in any other ways, and storage in data banks. Duplication of this publication or parts thereof is permitted only under the provisions of the German Copyright Law of September 9, 1965, in its current version, and permission for use must always be obtained from Springer. Violations are liable for prosecution under the German Copyright Law.

The use of general descriptive names, registered names, trademarks, etc. in this publication does not imply, even in the absence of a specific statement, that such names are exempt from the relevant protective laws and regulations and therefore free for general use.

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

Peter Bearman

Bluff Body Flow Research with Application to Road Vehicles3

David Schimel

Climate Change and the Energy Economy 15

Flow Field Characteristics

G. Iaccarino, F. Ham, Y. Khalighi, D. Bodony, P. Moin, B. Khalighi

Large Eddy Simulations and Acoustic Predictions in Automotive Applications 19

B. Khalighi, S. Jindal, J.P. Johnson, K.H. Chen, G. Iaccarino

Validation of the Immersed Boundary CFD Approach for Complex Aerodynamic Flows 21

B. Khalighi, J.P. Johnson, K.-H. Chen, R.G. Lee

Experimental Characterization of the Unsteady Flow Field behind Two Outside Rear-View Mirrors39

P. Merati, C.H. Leong, K.H. Chen, J.P. Johnson

Investigation of Bouyancy Driven Flow in a Simplified Full Scale Underhood – PIV and Temperature Measurements 53

K.H. Chen, J.P. Johnson, P. Merati, C.H. Leong

Investigation of Bouyancy Driven Flow in a Simplified Full-Scale Underhood – Numerical Study 75

Simon Watkins, Riccardo Pagliarella

The Flow Environment of Road Vehicles in Winds and Traffic 101

Separation Control for Drag Reduction

L. Taubert, I. Wygnanski

Preliminary Experiments Applying Active Flow Control to a 1/24th Scale Model of a Semi-Trailer Truck 105

<i>A. Seifert, O. Stalnov, D. Sperber, G. Arwatz, V. Palei, S. David, I. Dayan, I. Fono</i>	
Large Trucks Drag Reduction using Active Flow Control	115
<i>R. Spivey, R. Hewitt, H. Othman, T. Corke</i>	
Flow Separation Control on Trailing Edge Radii using Single Dielectric Barrier Discharge Plasma Actuators: An Application to Vehicle Drag Control	135
<i>L. Cattafesta, Y. Tian, R. Mittal</i>	
Adaptive Control of Post-Stall Separated Flow Application to Heavy Vehicles	151
<i>Jason Ortega, Kambiz Salari, Bruce Storms</i>	
Investigation of Tractor Base Bleeding for Heavy Vehicle Aerodynamic Drag Reduction	161
<i>C.N. Nayeri, J. Haff, D. Greenblatt, L. Loefdahl, C.O. Paschereit</i>	
Drag Reduction on a Generic Tractor-Trailer using Active Flow Control in Combination with Solid Flaps	179

Design Optimization Techniques Related to Vehicle Aerodynamics

<i>Ilhan Bayraktar</i>	
Advanced Aerodynamics and Cooling System Solutions for Higher Fuel Efficiency and Decreased Emissions	195
<i>Siniša Krajnović</i>	
Optimization of Aerodynamic Properties of High-Speed Trains with CFD and Response Surface Models	197
<i>Bhaskar Bhatnagar, Dan Schlesinger</i>	
Design Considerations for Maximizing Cooling Package Performance	213
<i>Clinton Lafferty, Kevin Horrigan, Ales Alajbegovic</i>	
Optimization and Correlation of a Class 8 Truck Cooling System	215

Train Aerodynamics

Alexander Orellano, Stefan Sperling

Aerodynamics Improvements and Associated Energy Demand Reduction of Trains	219
---	-----

Andreas Dillmann

The use of Aeronautical Experimental Facilities and Measurement Techniques for the Aerodynamic Investigation of High Speed Trains	233
--	-----

Sigfried Loose

Reduction of Skin-Friction Drag on a Generic Train Configuration	235
--	-----

Arnd Rueter

Head Pressure Effects of Trains and Locomotives – Engineering Calculation Approaches for Homologation Purpose	237
--	-----

Jing Zhao, Renxian Li

Numerical Analysis for Aerodynamics of High-Speed Trains Passing Tunnels	239
---	-----

Poster Session

Renxian Li, Jing Zhao, Shu Zhang

A Study of the Influence of Aerodynamic Forces on a Human Body near a High-Speed Train	243
---	-----

James C. Paul, Richard W. Johnson, Robert G. Yates

Application of CFD to Rail Car and Locomotive Aerodynamics	259
--	-----

Gandert M.R. Van Raemdonck, Michel J.L. van Tooren

Data Acquisition of a Tractor-Trailer Combination to Register Aerodynamic Performances	299
---	-----

Eddy Willemsen

Automotive Testing in the DNW-LLF Wind Tunnel	311
---	-----

Bruce Storms, Jason Ortega, Kambiz Salari

An Experimental Study of Tractor Base Bleed for Heavy Vehicle Aerodynamic Drag Reduction	317
---	-----

CFD, Numerical Methods and Application

Parviz Moin

Application of High Fidelity Numerical Simulations for Vehicle Aerodynamics	321
---	-----

Florian Menter

Scale-Adaptive Simulation in the Context of Unsteady Flow Simulations	323
---	-----

K. Sreenivas, B. Mitchell, S. Nichols, D. Hyams, D. Whitfield

Computational Simulation of the GCM Tractor-Trailer Configuration	325
---	-----

Ramesh Pankajakshan, Brent Mitchell, David L. Whitfield

Full-Scale Simulations of Drag Reduction Devices for Class 8 Trucks	339
---	-----

David Pointer, Tanju Sofu, Jimmy Chang, David Weber

Applicability of Commercial CFD Tools for Assessment of Heavy Vehicle Aerodynamic Characteristics	349
---	-----

Christopher J. Roy, Harshavardhan A. Ghuge

Detached Eddy Simulations of a Simplified Tractor/Trailer Geometry	363
--	-----

Kambiz Salari, Paul Castellucci

A Hybrid RANS/LES Turbulence Model for use in the Simulation of Turbulent Separated Flows	383
---	-----

Vehicle and Tire Spray and Vehicle Interaction

Simon Watkins

Spray from Commercial Vehicles: A Method of Evaluation and Results from Road Tests	387
--	-----

Charles Radovich, Dennis Plocher

Experiments on Spray from a Rolling Tire	403
--	-----

Florian Iser, Raimund A. Almbauer

Computational Simulation of the Flow Field of a Filter System inside Self-Ventilated Road Tunnels due to Heavy Vehicle Traffic	419
--	-----

B. Basara, S. Girimaji, S. Jakirlic, F. Aldudak, M. Schrefl

Experiments and Calculations Relevant to Aerodynamic Effects during Highway Passing Maneuvers	433
---	-----

Drag Reduction

Mark Page

Design & Test Techniques for Drag Reduction at Swift Engineering – A Racecar Manufactures Perspective	449
--	-----

Jason Leuschen, Kevin R. Cooper

Summary of Full-Scale Wind Tunnel Tests of Aerodynamic Drag-Reducing Devices for Tractor-Trailers	451
--	-----

Réjean Laflamme

A Fleet Operator's Perspective on Commercial Vehicle Drag Reduction	463
---	-----

Kenneth D. Visser, Kevin Grover

Class 8 Vehicle Fuel Savings using Sealed Single and Dual Open Aft Cavities	465
--	-----

Alec Wong, Kevin Horrigan

A Novel Approach to Heavy Vehicle Drag Reduction	467
--	-----

Linus Hjelm, Björn Bergqvist

European Truck Aerodynamics – A Comparison Between Conventional and CoE Truck Aerodynamics and a Look into Future Trends and Possibilities	469
--	-----

Mike Camosy, Andre Brown, Henri Kowalczyk, Gaylord Couthier

Advanced Experimental Methods for the Analysis and Aerodynamic Design of Heavy Vehicles	479
--	-----

Author Index	481
---------------------------	------------

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 vehicle flows than using the RANS equations coupled with a turbulence model.

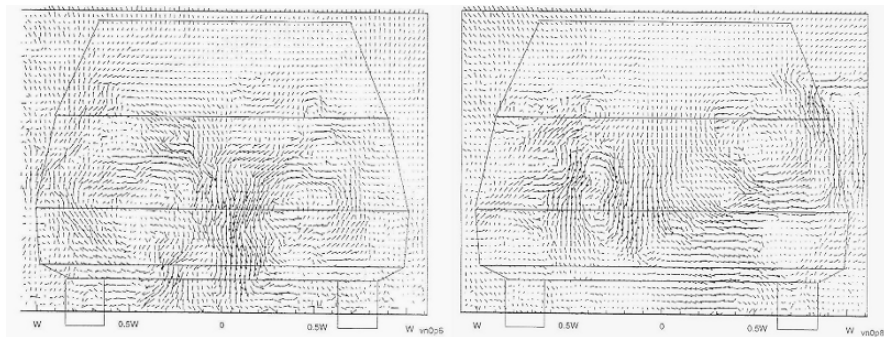


Fig. 1 Instantaneous cross-flow velocity maps measured in the near wake of a passenger car model at a vertical plane 20% of the car length from the rear bumper

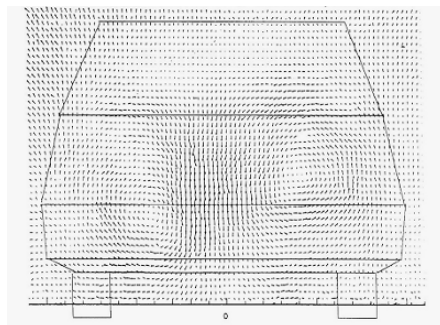


Fig. 2 The velocity field obtained from the average of only 10 instantaneous velocity maps

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.

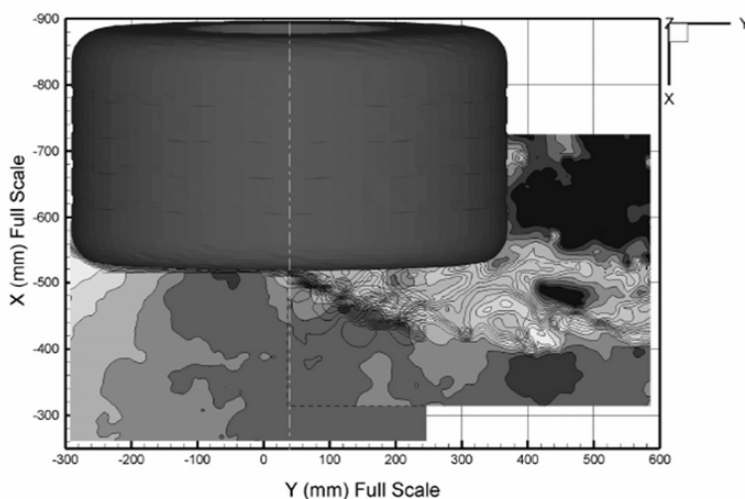


Fig. 3 An instantaneous velocity field measured in a horizontal plane just above the contact patch of a rotating wheel (Pegrum (2006))

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

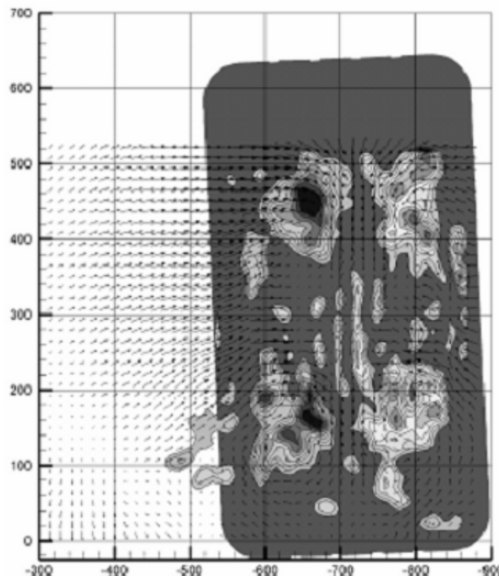


Fig. 4 Vorticity field showing the wake structure at 20% of a diameter behind a wheel (Pegrum (2006))

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.

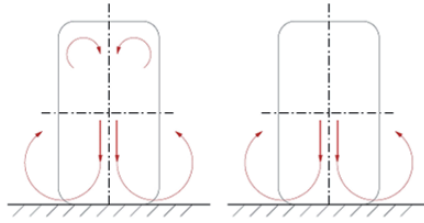


Fig. 5 Vortex Structure behind a Rotating Wheel, left 0.5 diameters downstream and right 2 diameters downstream (Saddington et al (2007))

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

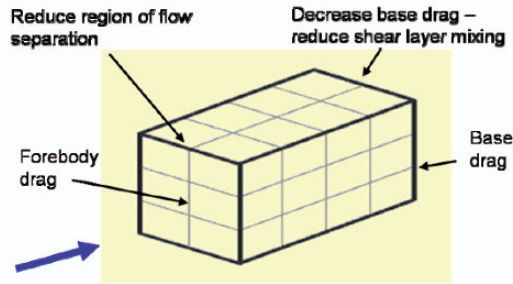


Fig. 6 A highly simplified heavy road vehicle

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.

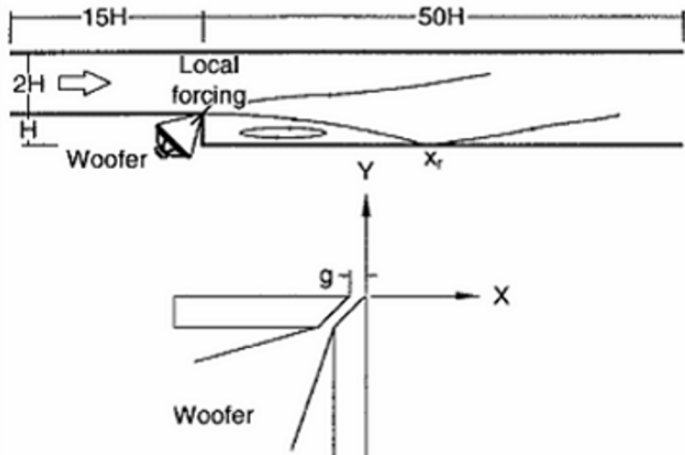


Fig. 7 Control of flow over a backward-facing step (Chun and Sung (1996))

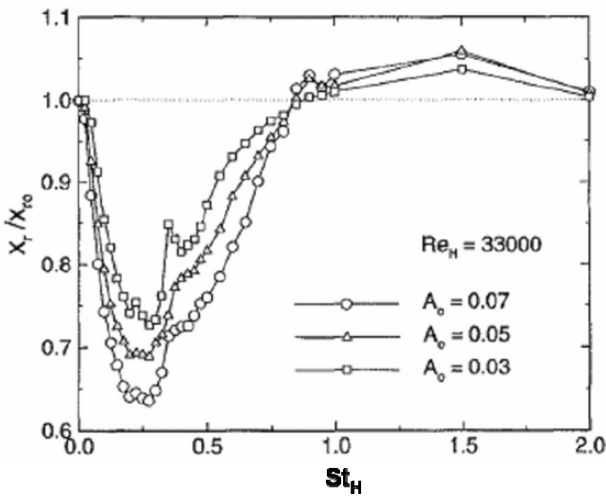


Fig. 8 Reattachment length behind a backward-facing step versus non-dimensional forcing frequency (Chun and Sung (1996))

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.

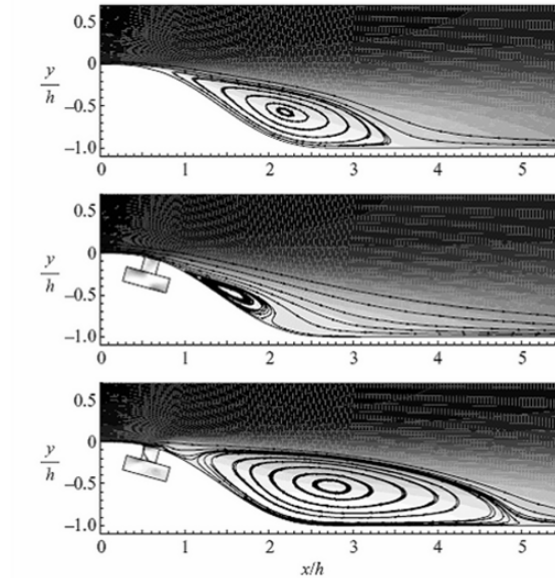


Fig. 9 Numerical simulations of flow control over a two-dimensional ramp using a massless jet: upper no control, middle low frequency forcing, lower high frequency forcing (Danois et al (2007))

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.

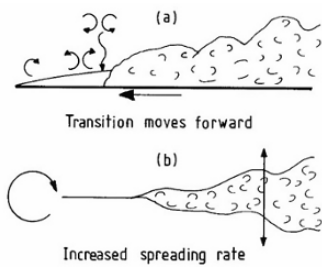


Fig. 10 Two mechanisms by which free stream turbulence affects flows (Bearman and Morel (1983))

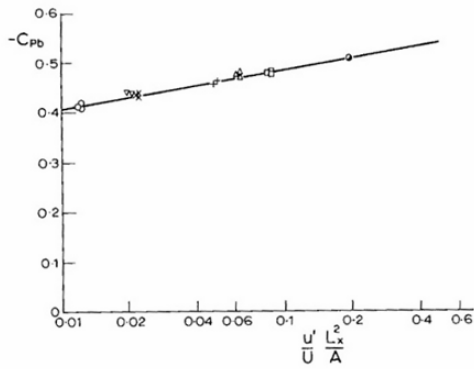


Fig. 11 Base pressure coefficients acting on square and circular thin plates set normal to turbulent flow

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, Skelthorne 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 CO₂ 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.

Large Eddy Simulations and Acoustic Predictions in Automotive Applications

G. Iaccarino¹, F. Ham¹, Y. Khalighi¹, D. Bodony¹, P. Moin¹, B. Khalighi²

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. Khalighi¹, S. Jindal¹, J.P. Johnson¹, K.H. Chen¹, G. Iaccarino²

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,

$$\mathcal{L}(\underline{U}) = 0 \text{ in } \Omega_f \text{ (volume)} \quad (1)$$

$$\text{with } \underline{U} = \underline{U}_r \text{ on } \Gamma_b \text{ (surface)} \quad (2)$$

Where $\underline{U} = (u, p)$ and $\mathcal{L}$ 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.

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

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 $(\text{Re})^{1.5}$ 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.

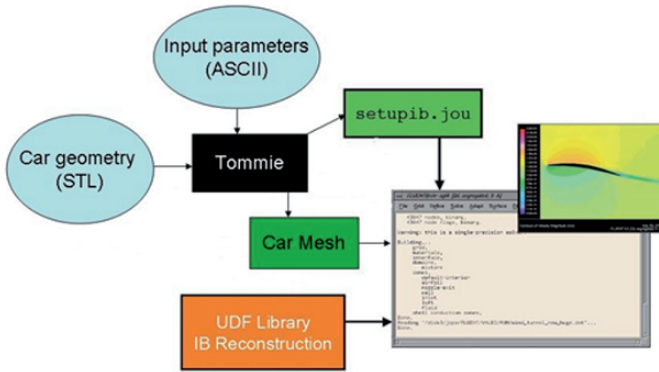


Fig. 1 Schematic diagram for the CAD to CFD solution in the IB approach.

1. TOMMIE

TOMMIE is the immersed boundary grid generator written in ANSI C++. It performs four main functions, namely the grid generation, the interface and the interior cell determination, wall-distance calculation and the evaluation of the weighting coefficients for the immersed boundary interpolation.

Geometries are imported in ASCII Stereo-Lithography format (STL). The STL representation of a surface is a collection of unconnected triangles of sizes inversely proportional to the local curvature of the original surface. The format is standard for the Rapid Prototyping community and all the CAD systems have the ability to export a given surface in the STL format automatically. This allows the treatment of any complex geometry without the need to generate an intermediate CFD surface mesh.

The geometry is placed on a structured background Cartesian mesh. This mesh covers the entire computational domain which includes the region inside the body. The generation of this grid is extremely simple and automatic. The mesh stretching is based on the location of the immersed boundary, and the cells are clustered near the object surface in accordance to the normal and tangential resolution as specified by the user. Several window refinements (rectangular, spherical and cylindrical) can be introduced to cluster the cells in the regions of interest such as the wake, underbody and the stagnation regions. The geometrical module then performs the separation (tagging) of the computational cells as shown in Fig. 2. The tagging procedure is based on a simple Ray Tracing (RT) technique. A random ray which originates from the location to be checked (grid nodes) is considered and the intersections between this ray and the given surface are computed. If the total number is even the point is outside the object. Similarly, if the total number is odd

the point is inside the object. The intersection between a ray (a 3D segment) and the surface (a collection of triangles) is carried out using the geometrical algorithms reported in [5]. In some cases, the RT could fail due to incorrect surface representations (e.g. unintentional missing triangles in the STL file). These gaps might be present in the original raw CAD representation or might arise while generating the STL representation of the CAD surfaces. To overcome this difficulty, before tagging any locations, three perpendicular rays are cast. In this case if the corresponding results are the same, the point is tagged, otherwise up to 20 additional random rays are traced and the most probable result is accepted. Alternatively, the STL file might be checked for inconsistency.

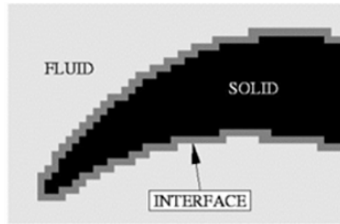


Fig. 2 Immersed Boundary tagging By Ray Tracing.

In the context of the IB technique, very refined grids are often required close to the highly curved immersed surfaces in order to properly represent the details of the geometry. Here a Local Grid Refinement (LGR) algorithm is used which allows an efficient clustering of cells. The present implementation is an extension of the classical Adaptive Mesh Refinement (AMR) technique for non-isotropic refinements. This requires a modification of the solution algorithm in order to handle grids with the hanging nodes. A detailed discussion of the implementation and the accuracy issues related to the present LGR method is given in [2, 6, and 7]. Another important aspect of the application of the LGR is the selection of the refinement/coarsening criteria. In the present implementation, the LGR is used to increase the resolution in the surroundings of the immersed boundary and, therefore, the only criterion used is the geometrical distance between each cell and the boundary. A Heavy-side tag function (using RT technique mentioned earlier) is used to mark the cells inside and outside the immersed body. An illustration of this procedure is shown in Fig. 3 for a geometry representing the letter F. Top row shows TOMMIE mesh, where a) to d) represents successive levels of refinement. The corresponding tagging function T is shown in the middle row, the dark area corresponds to the internal cells ($T = -1$) whereas the white area corresponds to the fluid cells ($T = 1$). The bottom row represents the numerical gradient of the Tagging function. Starting from a background mesh (Fig. 3a) and successively halving the cells until this gradient exceeds a prescribed value, the grid and the corresponding sharper representation in Fig. 3d is obtained. More detail about the LGR is available in [6, 7].

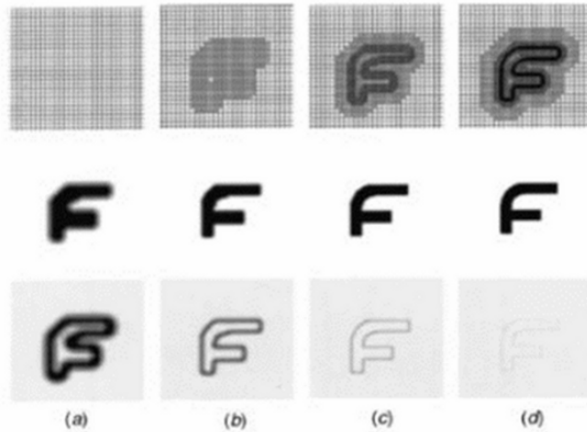


Fig. 3 Example of the automatic grid refinement strategy for immersed boundaries.

TOMMIE executes in batch mode and is extremely easy to use. A typical TOMMIE input file with its main features is reported in Appendix A.

The STLs can be added or deleted easily for the individual parts. Since there is no surface mesh associated with the volume cells, it amounts to saving time for the design/parametric studies. The surface mesh nodes need not to be aligned with one another and the neighboring surfaces need not have similar mesh sizes.

2. FLOW SOLVER

The flow solver used in this study is FLUENT V6.2. A user defined library (IBLIB) is used to handle the forcing functions in FLUENT. This library provides function hooks for reading/writing case files, initializing mesh, and implementing/updating wall boundary conditions.

The transport equations are solved only in the fluid cells, the solid cells are not considered and the values for interface cell values are obtained through interpolations. The steady incompressible 3D Reynolds-Averaged Navier-Stokes (RANS) equations are solved using a second-order implicit, cell-centered, finite volume scheme on Cartesian non-uniform grids. SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) procedure is used to obtain an intermediate velocity field that is corrected to divergence-free conditions using the solution of a Poisson's equation for the pressure.

Imposition of boundary conditions on the immersed surfaces is the key factor in developing an IB algorithm. It is also what distinguishes one IB method from other. Here a derivative of direct discrete forcing algorithm based on the ghost cell

approach is used [7, 8]. Local reconstructions of the solution in the vicinity of the immersed boundary are built for the interface cells (Fig. 4). Assuming nodal solution is available directly or through interpolation from cell-centers, Figs. 4b and 4c show the linear and quadratic 2D stencils. In the linear case, two exterior values (2 and 3) are used together with the wall value (0) to evaluate the solution close to the interface (1). For a quadratic interpolation (Fig. 4c), a generic flow variable Φ can be expressed as:

$$\Phi = C_1 n^2 + C_2 t^2 + C_3 n t + C_4 t + C_5 n + C_6 \quad (4)$$

where n and t are the local coordinates normal and tangential to the IB, respectively. The six coefficients then can be determined by using the five exterior points (2, 3, 4, 5 and 6) and the boundary condition point (0) which is the normal intercept from the node 1 to the IB. The word exterior here refers to the fluid side; therefore at node 1, the velocity is reversed in order to prevent the flow from penetrating and slipping on the immersed boundary. This approach can be considered as a generalization of the ghost cell approach where the boundary conditions are imposed by fixing suitable values of the solution outside the computational domain. Since polynomial interpolations are keen to introduce wiggles and spurious oscillations, an inverse distance weighted method is employed in the present study. A detailed treatment of the interface cells is described in [8]. Without IBLIB, Fluent would not have access to the velocity reconstruction for interface cells, and it would only obtain the solution over the stair-step approximation of geometry as prescribed by Tommie mesh.

Eddy viscosity is obtained through realizable k - ϵ turbulence model with enhanced wall treatment [9, 10]. Information needed for turbulence modeling and enhanced wall treatment, such as wall distances, is contained in Tommie mesh file and used through IBLIB. Currently only k - ϵ class of turbulence models is available for solving IB mesh in FLUENT.

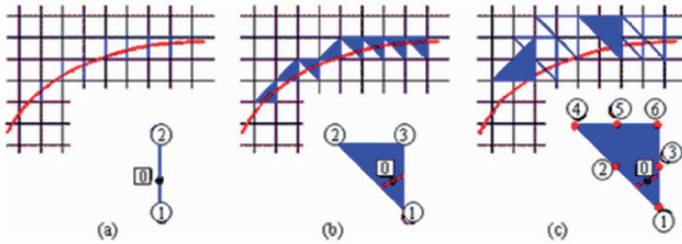


Fig. 4 Reconstruction stencils in the vicinity of the immersed boundary: (a) Linear 1D- scheme; (b) Linear 2D-scheme; (c) Quadratic 2D-scheme.

3. POST-PROCESSING

In the IB approach the immersed surface is disconnected from the computational grid and, therefore, it is not trivial to represent surface solution. Pressure values are projected from the stair-step grid onto the original body surfaces (STLs) using least-square interpolations.

The procedure used is shown in Fig. 5 schematically. Given a boundary node B and the surface normal in that location, the point P is defined at a certain distance δ from the surface. A least-square interpolation is used to obtain the velocity and the pressure at point P. The surface pressure at B is then obtained using the zero-normal gradient condition, $P_P = P_B$.

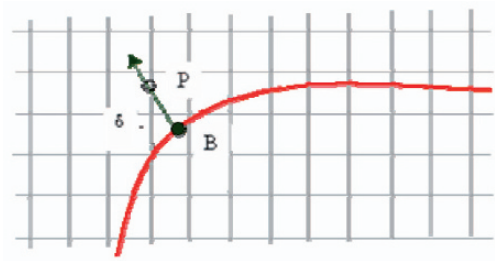


Fig. 5 Evaluation of surface quantities on the immersed body (Post-processing).

After each iteration, pressure is projected to the original STL surfaces automatically (transparent to the user) using the `DEFINE_ADJUST` function available in IBLIB. For a baffle (zero-thickness) surface, a shadow-copy is automatically made and pressure from one side of baffle is projected onto the shadow surface. The projected pressure is then integrated over the desired surfaces (original STLs) and resultant forces and moments could be calculated inside FLUENT framework as usual.

Similarly, calculation of the skin friction is not available directly in FLUENT. The cell-centered velocity gradients at wall surfaces are provided to FLUENT through IBLIB in order to calculate the skin friction.

TEST CASES AND RESULTS

1. FLAT PLATE

The plate dimensions considered for this investigation are 2m x 2m. The plate is placed perpendicular to the flow as shown in Figure 6a. The flat plate is specified with just two surface triangles in STL file. The surface mesh resolution does not matter as the volume mesh is made irrespective of the surface refinement. This case is important in the IB framework because it illustrates the IB support for baffle (zero-thickness) surfaces.

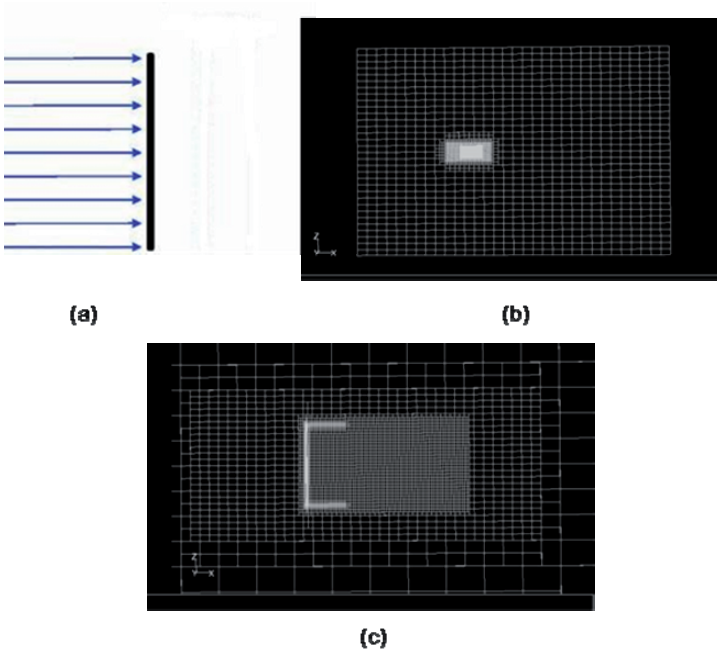


Fig. 6 (a) Schematic diagram of flow impinging on a squared flat-plate at 90° to the flow; (b) Symmetry plane view showing computational domain for flow over a perpendicular flat-plate; (c) Zoomed-in view of mesh near flat-plate.

The Reynolds numbers based on the frontal area (4m^2) and the free stream velocity (30m/s) is 8.2×10^6 . Mesh comprising of 150,000 cells is made with clustering around the wake and the stagnation regions. Figure 6b shows the symmetry plane for computational domain. A zoomed-in view near the plate and its wake is presented in Fig. 6c. Figure 7 shows the computed streamlines in the symmetry plane.

For the flat plate the experiments have shown that the separation points are fixed and the drag coefficient is 1.17 which is independent of Reynolds number [10]. The simulation result from the IB method is 1.18, which is in very good agreement with the experiments.

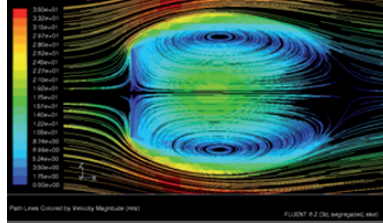


Fig. 7 Streamlines for flow over flat-plate perpendicular to flow direction.

2. BACKWARD STEP

Often the turbulent flow over a backward facing step is used to evaluate the accuracy of various numerical schemes. The main feature is a recirculation region just downstream of the step. The prediction of the reattachment length has proven to be challenging for numerical techniques. A brief review of incompressible flow over backward step can be found in [11, 12].

The step height (H) is 38.1 mm (1.5"), and the height of the inlet channel is 76.2 mm (3"). The uniform inlet velocity of 18.2 m/s is specified eight step heights ($8H$) upstream of the smaller inlet channel. The computational domain is extended about $29.3H$ downstream of the larger expansion channel.

A side-view of the computational domain is shown in Fig. 8. It is well known that the grid resolution in the near-wall and the high gradient regions is crucial for turbulent flow calculations. Two grids were tested for grid convergence study. The coarse grid is about $\sim 200 \times 10^3$ cells, and the fine mesh is about $\sim 1 \times 10^6$ cells. Both grids are generated with a $y^+ \sim 10$, but the fine grid is more refined in the recirculation region.

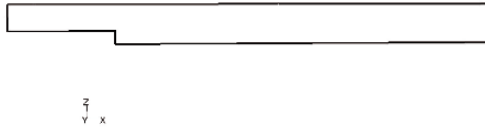


Fig. 8. Outline of the computational domain for backward step.

Figure 9 shows both the meshes and their respective recirculation regions. The experimental data for the length of recirculation zone is $7.1 \pm 0.5H$ [11, 12]. The simulations reveal a recirculation zone length of approximately $7.03H$ for the

coarse mesh and $7.11H$ for the fine mesh. It is clear that, both meshes capture the essential features of the flow in terms of recirculation bubble length and are in very good agreement with the experiment.

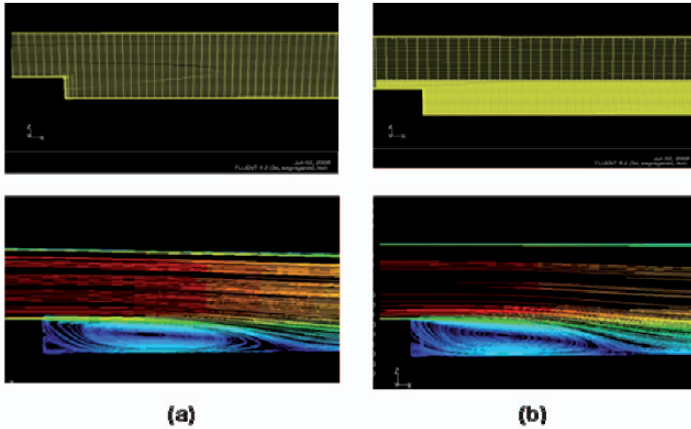


Fig. 9 Side-view of the mesh (top) and the streamlines (bottom). The colors represent the velocity magnitudes: (a) Coarse mesh; (b) Fine mesh

3. THE SEDAN

A picture of a sedan (97 Pontiac Grand-Prix) used in this study is shown in Figure 10. The length of the model is $\sim 5\text{m}$ (x axis), the width is $\sim 2\text{m}$ (y-axis) and the height is $\sim 1.4\text{m}$ (z-axis). A coordinate system is attached to the front tip of the vehicle in the symmetry plane. The water-tight car surface was exported in STL format from HYPERMESH with overall 270,800 surface triangles. The geometry was distributed over 84 STLs including 18 baffles. Four fluid zones were generated in TOMMIE, one each for the outer body, the radiator, the condenser, and the fan.



Fig. 10 1997 Pontiac Grand-Prix

The model is placed in a wind tunnel test-section with a cross-section of $2.1L \times 1.1L$, where L is the length of the vehicle. The length of the test section is about $4.3L$. Figure 11 shows the symmetry plane of the computational domain depicting the location of the car with respect to the wind-tunnel. A free-stream velocity of 22.352 m/s is specified at the inlet with a low turbulence intensity of 0.6% . The Reynolds number based on the vehicle length and the free stream velocity is 7.6×10^6 . The outlet is specified as pressure outlet and the wind-tunnel walls are treated as no-slip walls. Heat exchangers are treated as porous media and the fan-outlet surface is specified as momentum-jump based on the normal velocity (Fan BC in FLUENT).

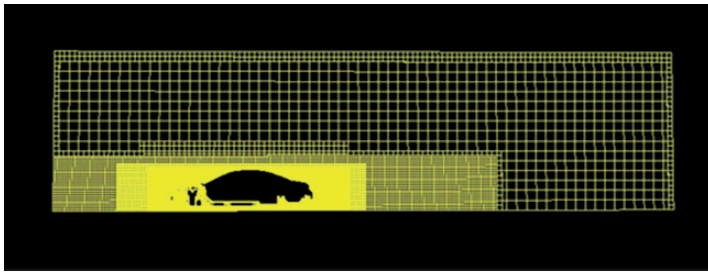


Fig. 11 Symmetry-plane view of the wind-tunnel test section for the sedan.

Three different IB meshes are considered for investigating the effect of grid refinements. These meshes, referred to henceforth as “IB1”, “IB2”, and “IB3”, respectively, differed only in the number of cells used for the simulations. Case IB1 contains about 12 million cells, IB2 has about 16 million cells, and IB3 is about 17 million cells. TOMMIE requires about 4 CPU hours and 7 GB of RAM for IB1 case (12 million) on an IBM p655 workstation. The IB simulations were compared to the conventional body-fitted approach. For the body-fitted approach two different simulations of 7 (BF1) and 10 million (BF2) cells were carried out. TGRID was used for the volume meshing in the body-fitted cases. An image of the mesh in the symmetry plane for cases IB3 and BF2 (body-fitted) are shown in Fig. 12.

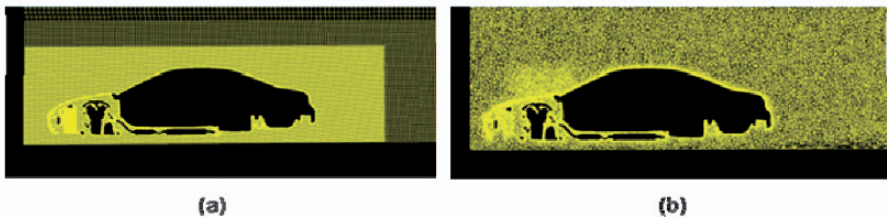


Fig. 12 Side-view showing mesh density near the sedan and its wake: (a) IB1; (b) BF2

The residuals for each of the flow variables were monitored and solution convergence was achieved when the residuals were down to approximately 10^{-5} . Figure 13 shows the convergence pattern for both simulations. Convergence of IB meshes per iteration seems to be similar to that of the regular body-fitted meshes.

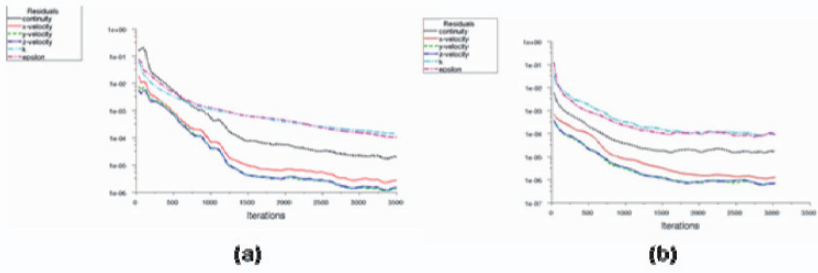


Fig. 13 Relative FLUENT residuals for flow variables depicting convergence: (a) IB3; (b) BF2

Figure 14 shows the velocity contours for IB (IB3) and body-fitted (BF2) meshes in the symmetry plane. Both simulations show similar qualitative features. There is a stagnation region in front of the vehicle, the flow then accelerates on the hood and stagnates at the wind-shield bottom. It then strongly accelerates on the front wind-shield. There is some pressure recovery on the top of the cabin. The flow then separates at the edge of trunk. The underbody flow also accelerates which is due to less area caused by wheels and the underbody components. There is a small wake at the rear end characterized with a strong downwash.

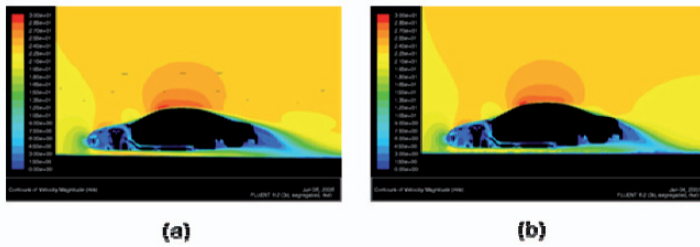


Fig. 14 Velocity contours near the sedan and its wake: (a) IB3; (b) BF2

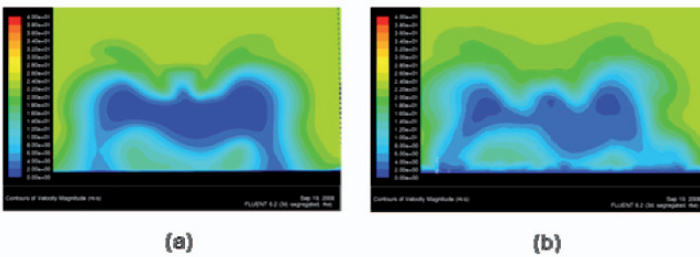


Fig. 15 Cross plane velocity contours in the sedans near wake at $x = 6m$: (a) IB3; (b) BF2

Figure 15, shows the velocity contour plots in the near wake at an axial plane of $x = 6m$. The flow looks symmetric which is expected. Also both simulations seem to have a similar flow structure in the near wake.

Pressure coefficients (C_p) are calculated using the equation,

$$C_p = (P - P_{ref}) / (0.5 \times \rho \times U_\infty^2) \quad (5)$$

where P_{ref} is the reference static pressure measured at the inlet of the computational domain (wind-tunnel), P is the surface pressure, ρ is the air density and U_∞ is the free-stream velocity.

Figure 16 shows the comparisons between the two simulated pressure distributions (C_p) on the symmetry plane. The overall agreement between the body-fitted and the IB simulations is very good. However, peaks in the separation region are somewhat different. This could be due to the absence of prism layers or the sufficient grid resolution near the wall surfaces in IB meshes. Experimental pressure measurement for this vehicle is not available.

Table 1 summarizes the experimental and the computed drag coefficients for this vehicle. The values are normalized by the experimental value. The result for IB3 simulation case is in very good agreement with the experimental results.

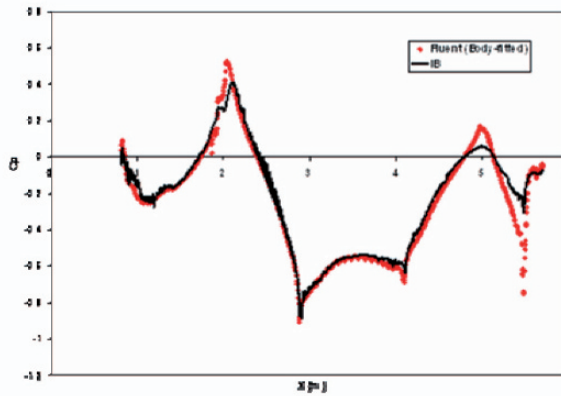


Fig. 16 Pressure distribution comparison on the top of the sedan (symmetry plane).

Table 1 Drag coefficient comparison for the sedan.(Drag coefficients are normalized by the experimental value)

Sedan	Case	Drag Coefficient
Experiment	-	100
IB Simulation	IB1 – 12M cells	101
	IB2 – 16M cells	99
	IB3 – 17M cells	100
Body-fitted Simulations	BF1 – 7M cells	95.3
	BF2 – 10M cells	94.4

4. THE SUV

A picture of the Chevy Tahoe SUV is shown in Fig. 17. The water-tight geometry was exported from ANSA comprising of 923,409 STL facets. A free stream velocity of 31.11 m/s is specified at the inlet of wind-tunnel. All other boundary conditions are similar to the sedan case.

**Fig. 17** 2006 Tahoe SUV.

The IB mesh consists of 13 million cells, and grid generation took about 4 hours on an IBM workstation. Results are compared to a body-fitted solution-adapted (based on pressure gradient) mesh of 8 million cells. A view of both grids in the symmetry plane is shown in Fig. 18.

Figure 19 shows the velocity contour plots for both the IB and the body-fitted cases. The qualitative features of the flow are similar in both simulations. The recirculation region is slightly larger in the IB case which could be due to the less resolved grid in the wake.

Figure 20 shows the wake streamlines in the symmetry plane. The main features of the flow are the shear layers originating at the upper and lower edges of the model. A strong reversed flow region is formed behind the model which is bounded by these two shear layers. There is a rapid upward deflection of the underbody flow and a strong upsweep in both the computed fields. The upper shear layer in the body-fitted seems to be somewhat thicker than the IB simulations.

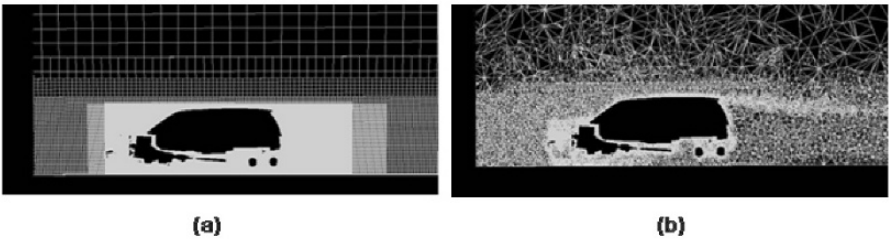


Fig. 18 Side-view showing mesh density near the SUV and its wake: (a) IB; (b) Body-fitted

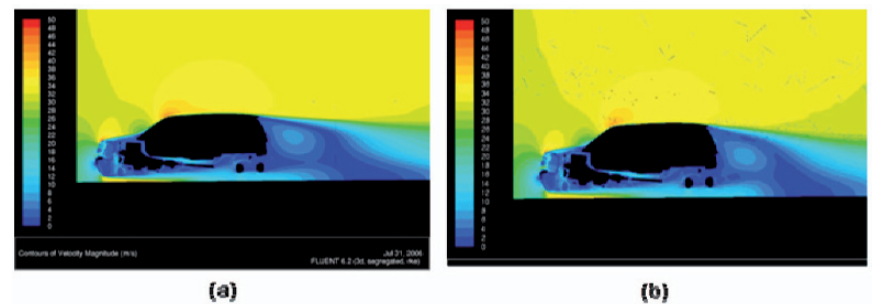


Fig. 19 Side-view showing velocity contours near the SUV and its wake: (a) IB; (b) Body-fitted

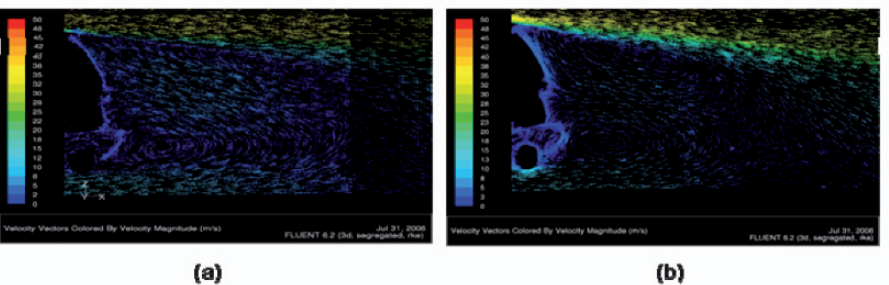


Figure 20. Side-view showing velocity vectors for the SUV in its wake: (a) IB; (b) Body-fitted

Pressure coefficient distributions are presented in Fig. 21. The overall agreement between the two simulations (IB and body-fitted) is very good.

Comparison of drag coefficients for the SUV is summarized in Table 2. The results show that the IB method does a very good job in predicting the drag coefficient.

Table 2 Drag coefficient comparison for the SUV. (Drag coefficients are normalized by the experimental value)

SUV	Drag Coefficient
Experiment	100
IB simulation	97.3
Body-fitted simulation	93.5

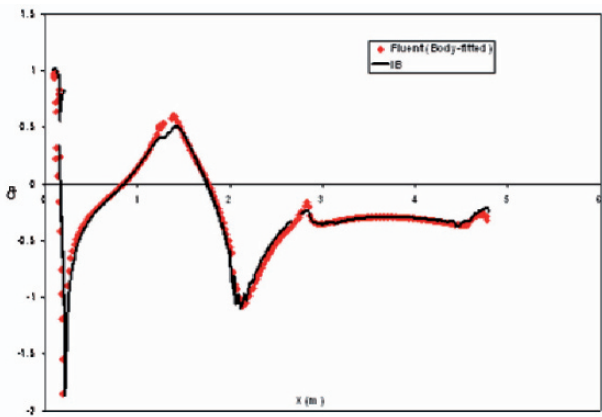


Fig. 21 Pressure distribution comparison on the top of the SUV (symmetry plane).

Conclusions and Future Work

The IB method was successfully validated for flat-plate and backward step geometries. Further, to test its applicability for standard aerodynamics applications in automotive industry, it has been applied to the simulation of the flow past a sedan and an SUV. For the sedan and SUV applications the STL surfaces were obtained from the CAD geometry that had already undergone clean up operations in preparation for the FLUENT body-fitted solutions. These STL files therefore represented “water-tight” surfaces prior to their input to TOMMIE. This was done to compare solver accuracy when the identical surfaces are used for both the IB and body-fitted utilization of the FLUENT solver. The results are in good comparison. Convergence per iteration is similar to traditional body-fitted methods. However, for similar accuracy IB results requires more number of cells. This is due to IB’s inability to generate high aspect-ratio cells near angled wall surfaces. The worst case being a surface at 45 degrees to horizontal, for which IB refines in tangential

directions as much as it does in normal direction, thereby increasing the overall number of cells [4].

Future work will study the usage of STL files exported directly from a vehicle program's CAD database with no or minimal CAD clean up operations, sometimes referred to as "raw" STL files. Since CAD clean up operations comprise the majority of time spent executing a typical body-fitted aerodynamics CAE study, a successful validation of the IB method's capability to calculate accurate results using raw STL files will be a significant milestone in reducing project execution times. This in turn will improve the capability of aerodynamic CAE to impact vehicle design decisions.

It is also intended to investigate the IB method for solving coupled flow and thermal problems.

References

1. Ahmed S, "Computational Fluid Dynamics," in *Aerodynamics of Road Vehicles*, 4th ed., Editor: Hucho, W.-H., 1998.
2. Jindal S, Khalighi B and Iaccarino G, "Numerical Investigation of Road Vehicle Aerodynamics Using Immersed Boundary Approach", SAE paper no. 2005-01-0546.
3. Peskin C, "The Fluid Dynamics of Heart Valves: Experimental, Theoretical and Computational Methods," *Annual Review of Fluid Mech.*, 14:235-59.
4. Mittal R and Iaccarino G, "Immersed Boundary Methods," *Annual Review of Fluid Mech.* 2005, 37:239-61.
5. O'Rourke J, Computational geometry in C, *Cambridge university press*, 1998. Durbin P and Iaccarino G, "An approach for local grid refinement of structured grids," *Journal of Computational Physics*, pp. 639-653, 2002.
6. Iaccarino G, Kalitzin G, Moin P and Khalighi B, "Local Grid Refinement for an Immersed Boundary RANS Solver", AIAA paper No. 2004-0586, 2004.
7. Iaccarino G and Verzicco R, "Immersed Boundary Technique for Turbulent Flow Simulations," *Applied Mechanical Review*, pp. 331-347, 2003.
8. Kalitzin G and Iaccarino G, "Turbulence Modeling in an Immersed Boundary RANS Method", *CTR Annual Briefs*, 2002.
9. Iaccarino G, Kalitzin G and Khalighi B, "Towards an Immersed Boundary RANS Flow Solver", AIAA paper No. 2003-0770, 2003.
10. Aerodynamic Drag, Data for Airfoils, Wings, Aircraft, Automobiles, <http://www.aerodyn.org/Drag/tables.html>
11. Kim J, Kline S and Johnson J, "Investigation of a Reattaching Turbulent Shear Layer: Flow Over a Backward-Facing Step," *Journal of Fluid Engineering*, 102, pp. 302-308, 1980.
12. Chen K and Liu N, "Evaluation of a Non-Linear Turbulence Model Using Mixed Volume Unstructured Grids," AIAA paper No. 98-0233, 1998.

Experimental Characterization of the Unsteady Flow Field behind Two Outside Rear-View Mirrors

Bahram Khalighi¹, James P. Johnson¹, Kuo-Huey Chen¹, Richard G. Lee²

¹ General Motors, Warren, Michigan, USA

² NRC Institute for the Aerospace Research, Ottawa, Ontario, Canada

bahram.khalighi@gm.com

Abstract The unsteady flow field behind two outside rear-view automobile mirrors was examined experimentally in order to compile a comprehensive database for the validation of the ongoing computational investigation effort to predict the aero-acoustic noise to the outside rear-view mirrors. This study is part of a larger scheme to predict the aero-acoustic noise due to various external components on automobiles. To aid with the characterization of this complex flow field, mean and unsteady surface pressure measurements were undertaken in the wake of two mirror models. Velocity measurements with particle image velocimetry were also conducted to develop the mean velocity field of the wake. Two full-scale mirror models with distinctive geometrical features were investigated.

1 Introduction

The unsteady flow field in the wake of external rear-view mirrors of a typical production car is known to be a significant source of aerodynamic noise inside and outside of a vehicle in motion. The shape of the mirror assembly, its attitude in relation to the flow and the adjacent windowpane, the details of its installation on the car, and the relative airspeed are all factors that can influence the aero-acoustics of automobiles. Because the wake and the adjacent shear layers around a mirror are an important source of noise, a numerical study of the flow field around and in the wake of typical external side-view mirrors was initiated as a collaborative project between General Motors and the Center of Turbulence Research, Stanford University. Experimental validation of the predictions is necessary and it is intended to include the characterization of the steady and unsteady pressures in the wake of two mirrors, as well as direct measurements of acoustic radiation in a well-controlled experiment that could be reproduced easily in the computational domain. These validation measurements were organized in a two-phase wind-tunnel test program: The first phase focused on mean pressure and unsteady

pressure measurements and mean flow field measurements of the mirror wake; the second phase will concentrate on the measurement of the acoustic radiation. This paper describes the first phase of the validation program.

2 Experimental Details

The characterization of the unsteady flow field in the wake of the rear-view mirrors was performed in the 0.9m Pilot Wind Tunnel at the NRC Institute for Aerospace Research (NRC-IAR) in Canada. Each mirror model was mounted at zero-yaw angle on a ground plane model near the nozzle inlet of the test section (Fig. 1). An array of static pressure taps and fast-response pressure transducers (microphones) were located on the ground plane in the wake of the mirrors; additional pressure taps were located on the floor upstream of the mirror and on the reflective surface of the mirror. A two-dimensional particle image velocimetry (PIV) system captured the streamwise (u) and the lateral (v), or vertical (w) velocity components in the wake. The mean or time-average velocity field was determined for three horizontal planes (u - and v -components of velocity and three vertical planes (u - and w -components).

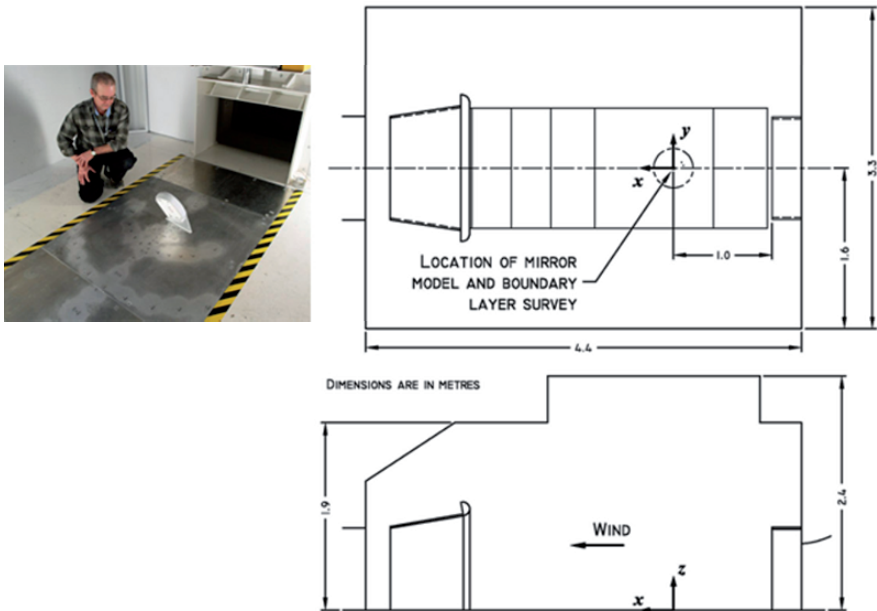


Fig. 1 The layout of a mirror model on the ground plane in the Pilot Wind Tunnel and a schematic of the plenum.

0.9m Pilot Wind Tunnel

The 0.9m Pilot Wind Tunnel at the NRC-IAR is a closed-return facility featuring a $\frac{3}{4}$ - open-jet test section enclosed within a large plenum that is 4 m long, 3 m wide, and 2 m high (Fig. 1). The cross-sectional dimensions of the nozzle exit are 1 m wide by 0.6 m high, and the highest velocity attainable in the test section is approximately 50 m/s. The freestream flow has longitudinal and vertical turbulence intensities of 0.8% and 1.3%, respectively; the displacement thickness of the boundary layer is estimated to be 4 mm at a velocity of 30 m/s.

Measurements of the background noise level for an empty test section were acquired with a microphone placed outside the flow of the test section in a quiescent corner of the plenum enclosure. The sound pressure level reaches a level of approximately 98 dB at 30 m/s, the speed at which the majority of testing was performed. No steps were taken in this experiment to insulate the test section against the noise emanating from the fan and the drive motor. An acoustic resonance in the proximity of 14.5 Hz was known to persist in the plenum. Because the resonance was found to be especially significant at a velocity of 35 m/s, the decision was made to conduct the pressure and PIV measurements at 30 m/s.

The coordinate system referred to in this paper is defined in Fig. 1. The x-axis is aligned with the freestream flow, the y-axis is normal to the flow in the lateral direction, and the z-axis is normal to the floor of the plenum.

Mirror Models

The full-scale mirror models that were used in the tests are shown in Fig. 2.

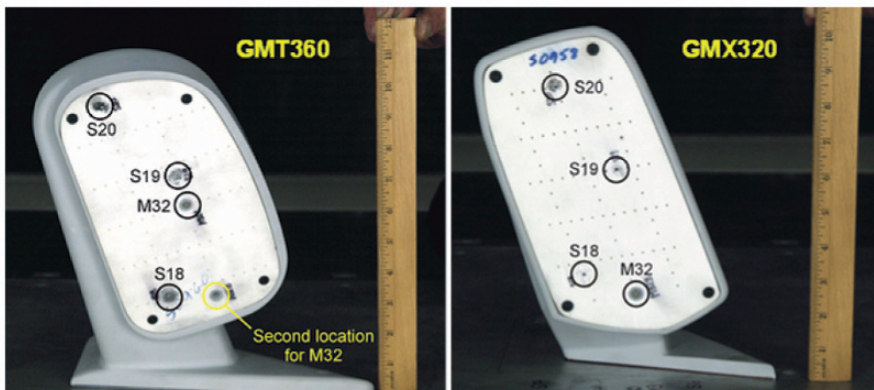


Fig. 2 The mirror models and the layout of static pressure taps and an unsteady pressure transducer on the reflective surface of each model.

The model of the GMT360 mirror is generally found on sport-utility vehicles, features distinctly rounded edges on its windward surface, and is the largest of the mirrors. In contrast, the smaller GMX320 mirror is typically found on the Cadillac brand and has sharper edges on the windward surface of its body. Each body features a cavity that is covered by a model of the reflective surface. For pressure measurements, this reflective-surface model was a thick plastic plate perforated with an array static pressure taps, from which only a few were actually used. For PIV measurements, the reflective-surface model was a simple imperforated aluminum sheet; moreover, the entire surface of each mirror model was painted flat black to avoid undesirable reflections of laser light. The mirrors were mounted on the ground plane model (e.g., the floor of the plenum), which represents the adjacent door window of an automobile.

The location of each mirror model on the ground plane corresponded with the origin of the x - y coordinate system shown in Fig. 1. This particular model location – being 1 m from the nozzle exit – was selected because it falls within a certain range of longitudinal positions for which blockage corrections to the dynamic pressure are known to be unnecessary.

Mean Pressure Measurements

Measurements from twenty static pressure taps were acquired. Fifteen static pressure taps were laid on the ground plane along the centerline of the test section – three taps were located upstream of the mirror model and twelve taps were placed in its wake (Fig. 3).

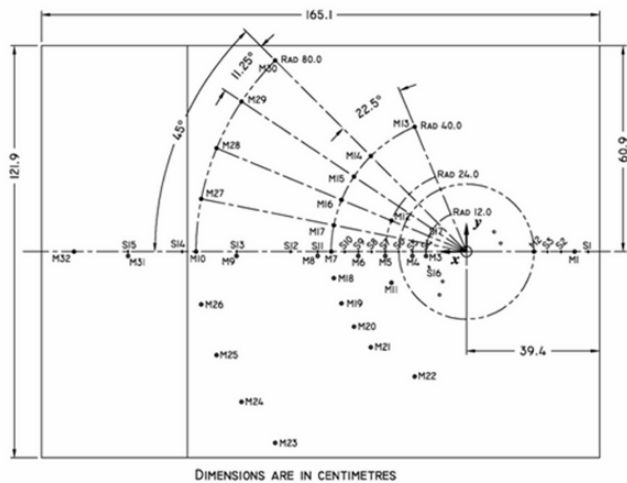


Fig. 3 The layout of fourteen static pressure taps (S) and thirty-one unsteady pressure transducers (M) on the ground plane

Two additional static pressure taps were placed in the wake off the centerline and a further three were installed on the reflective-surface model of the mirror (Fig. 2).

Static pressures were measured with two Scanivalve electronic scanning pressure modules, which have a differential pressure range of 2.5 kPa (10 inches of water) and an accuracy of 5 Pa. Mean pressures were derived from static pressure data acquired at a rate of 100 samples/second for a period of 10 seconds.

Unsteady Pressure Measurements

Measurements of unsteady pressure with sensitive microphones in the wake of a bluff body have been successfully demonstrated, for example, in the case of a two-dimensional backward-step (Lee and Sung, 2002). In the investigation at the NRC-IAR, Brüel & Kjær Delta Tron pressure-field ¼-inch pre-polarized microphones (Type 4944A) were employed to measure the unsteady surface pressures in the wake of the mirror models. The microphones are vented for pressure equalization, have a flat pressure-field response from 4 Hz to 70 kHz, and a dynamic range of 30 to 170 dB(A). With protective nose cones removed, thirty-one microphones were flush-mounted with custom fittings in the ground plane (Fig. 3). Of the microphones mounted along the streamwise centerline of the ground plane, some were collocated with static pressure taps. An additional microphone was installed in the reflective face of the mirror model. Because of the depth of its cavity, two locations on the reflective surface of the GMT360 mirror were possible (Fig. 2); however, only one location was feasible on the GMX320 mirror because of its shallower cavity. Unsteady pressure data was sampled at a rate of 44 kHz for a period of 70 seconds, yielding a frequency bandwidth of at least 20 kHz.

Particle Image Velocimetry

Three measurement planes were selected for each view. Two overlapping image segments of the measurement plane covered the near-wake and the far-wake (a distance of approximately 550 mm in the streamwise direction) and were adequate to resolve most of the recirculation zone, the freestream, and the shear layer in between. The elevation of the views was concentrated on the wake of the main body of the mirror; the wake of the neck was not surveyed. Two hundred double-frame images (the limit that could be stored in the random access memory of the PIV computer) were typically acquired for each segment of the wake. Details of the PIV recording parameters are listed in Table 1. INSIGHTTM, the software package developed by TSI, was used to acquire and analyze the PIV image data. The *deformation grid* scheme – a particle-displacement analysis algorithm developed by TSI for application to high shear flows – was employed to compute the particle displacements in the images. Although it was general practice to acquire the maximum number of images possible, not all were required

by the particle-displacement analysis algorithm to render a satisfactory mean flow field. The number of images processed varied. In the first segment, which covered the near-wake, only seventy-five images were necessary to yield a satisfactory mean flow. In the second segment farther downstream, where high shear flow features were less predominant, twenty-five images were processed instead. Within the total observed flow field, approximately 8,500 velocity vectors were resolved at a nominal spatial resolution of 3.5 mm.

Table 1 PIV recording parameters for mirror wake measurements

Flow geometry	$V_\infty = 30$ m/s parallel to the light sheet
Maximum in-plane velocity	$V_\infty = 35$ m/s
Field of view	355 mm $\times$ 267 mm (W $\times$ H)
Interrogation volume	7 mm $\times$ 7 mm $\times$ 1.5 mm (W $\times$ H $\times$ D)
Observation distance	x-y plane: 859 mm x-z plane: 804 mm (GMT360), 940 mm (GMX320)
Recording method	15 Hz double frame/single exposure
Recording medium	2-megapixel, full frame CCD (1600 $\times$ 1200 pixels)
Recording lens	$f = 28$ mm, f-stop = 1.4
Illumination	Nd:YAG laser, 120 mJ/pulse
Pulse delay	x-y plane: $\Delta t = 12.5$ μ s x-z plane: $\Delta t = 12.5$ μ s
Seeding material	olive oil

3 Results and Discussion

Results for mean pressure, unsteady pressure, and the mean velocity field are presented for 30 m/s. Further results for mean pressure are presented for 14, 22, and 39 m/s.

Mean Pressure

The variation of static pressure coefficient (C_p) on the centerline of the ground plane against streamwise distance is plotted in Fig. 4 for both mirror models and the baseline case (i.e., with no model in place) over four wind speeds. The results reflect the typical streamwise mean pressure distribution fore and aft of a bluff body. Upstream of the models, the increase of positive C_p reflects increasing stagnant flow due to the presence of the mirror. Downstream of the models, the results show two distinct features of the flow field, namely a wake and the attachment of the local flow. The presence of significant negative C_p downstream of the leeward face of each mirror reflects the wake of the model. At the streamwise distance where $C_p = 0$, the flow attaches to the ground plane and the recirculation region

closes. By this indication, the wake of the GMT360 mirror is slightly longer than that of the GMX320, although the latter generally has a higher negative peak in C_p . The fact that GMT360 is taller by 24 mm may partly account for its longer wake. The results for the wake of GMX320 collapse very well over the four wind speeds; for GMT360, however, the variations in C_p suggest a Reynolds number dependence, which may be attributed to the rounded features of its windward surface. Over the reflective surface of GMT360 the base pressure is reasonably constant, whereas for the GMX320 the base pressure varies significantly over the face.

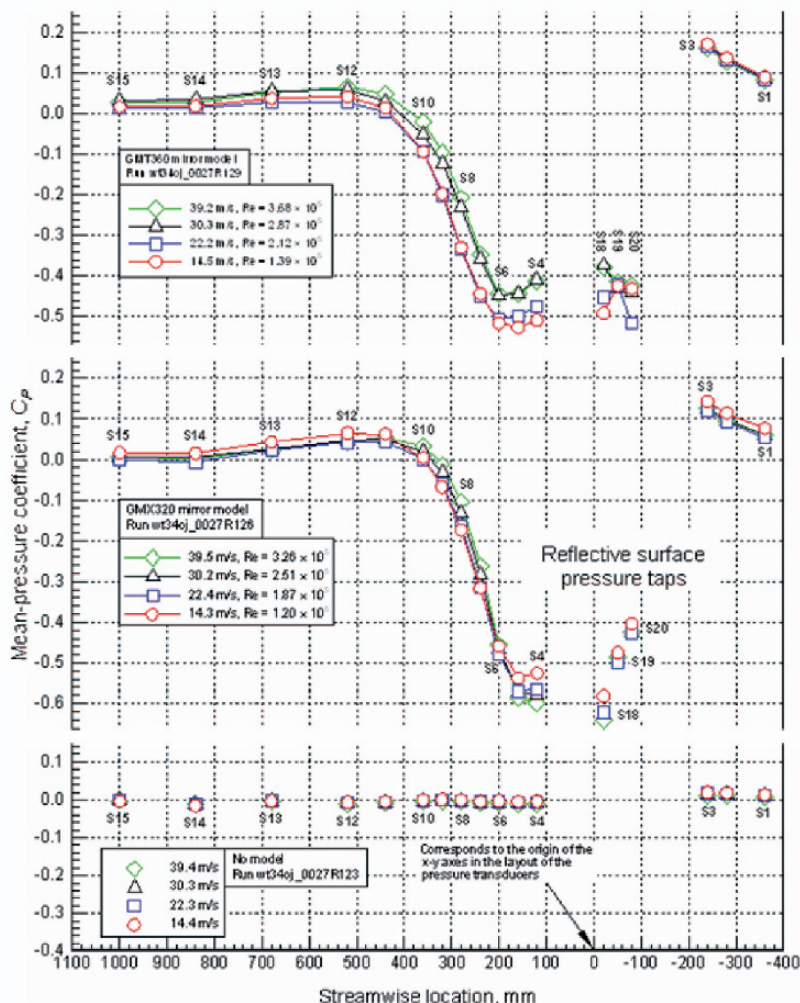


Fig. 4 Mean-pressure coefficient on the surface of the ground plane plotted against streamwise-centerline location for both mirror models and the baseline case.

Unsteady Pressure

Figures 5 and 6 present typical examples of mean-square spectral distributions (S_{C_p}) and coherence functions for both mirror models at a wind speed of 30 m/s. The spectra were derived from data acquired with the microphones embedded in the ground plane and were computed from an average of ninety-four 32,768-point fast-Fourier transforms of the unbiased time-histories of pressure. All plots of spectra and coherence have a frequency resolution of 1.3 Hz. For the coherence plots, correlations were developed with respect to the farthest downstream microphone, e.g., microphone M31 from the line of microphones that parallels the streamwise centerline (Fig. 3).

The mean-square spectral distributions and coherence functions for the GMT360 mirror model are shown in Fig. 5. In Fig. 5a, the results presented are for microphones that are located on the streamwise centerline of the ground plane with the exception of microphone M32, which was located on the reflective surface of the mirror (Fig. 2). In Fig. 5b, the microphones are located off the centerline. In both sets of results, the mean-square spectral distributions are dominated by relatively low-frequency turbulent excitation and sharp peaks are clearly evident in the spectra at a frequency of 45.6 Hz (with the exception of microphones M1 and M3). These features are also reflected in the spectra of the corresponding microphones on the opposite side of the centerline. The corresponding Strouhal number (St) is 0.212, based on the width of the mirror body. The results suggest the presence of an alternating trailing vortex system. The case for trailing vortices is substantiated in Fig. 7c, which shows that a good correlation exists between the centerline microphone M31 and M10/M9/M8/M6

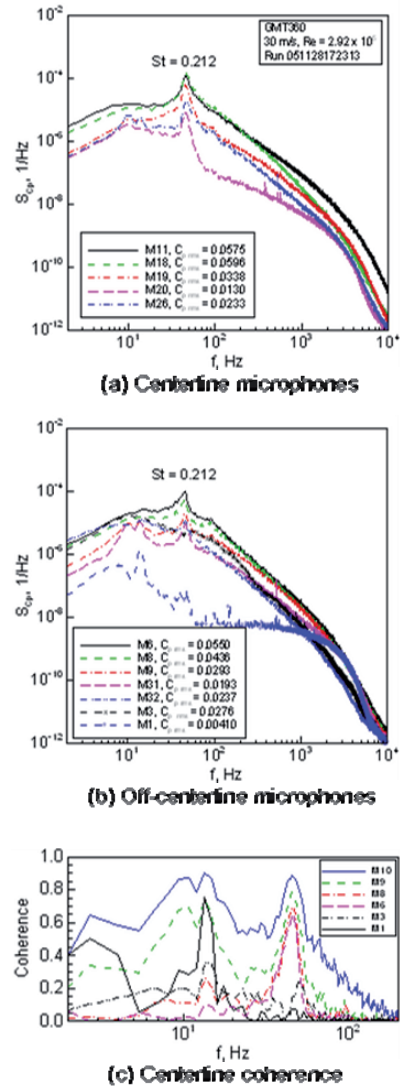


Fig. 5 The mean-square spectral distributions of C_p and coherence functions for the GMT360.

at a frequency of 45.6 Hz, whereas the correlation between M31 and M1 (located upstream of the model) at the same frequency is weak. The fact that centerline microphones in the wake of the mirror are registering the alternating trailing vortices serves as an indication of the size of these flow structures, which emanate from the sides of the mirror. On another note, a reasonable coherence is observed between M31 and M10/M9/M1 at 13.4 Hz, suggesting that these microphones registered the acoustic resonance in the plenum. In contrast, the coherences between M31 and M8/M6/M3 at 13.4 Hz are relatively weak, an indication perhaps that M8/M6/M3 either lie inside or are in close proximity to the recirculation zone of the wake. If so, then the energy of the turbulent excitation within the recirculation zone dominates the energy of the acoustic resonance.

The mean-square spectral distributions and coherence functions for the GMX320 mirror model are illustrated in Fig. 6. The spectra shown in this figure are for the same microphones appearing in Fig. 5. In Fig. 6a the 45-Hz peak identified previously in the GMT360 results, is conspicuously absent in the spectra for the centerline microphones in the wake of GMX320. However, the spectra for the off-centerline microphones, appearing in Fig. 6b, reveal broad shallow peaks in the proximity of 45 Hz. Again, there exists a degree of symmetry in the spectra for corresponding microphones on both sides of the centerline. It is suggested that these results may be the signature of an *Arch*-type trailing vortex, which sheds as one feature from the GMX320 unlike alternating trailing vortices (Okuda and Taniike, 1988). If in fact a trailing vortex sheds from GMX320, then the reasonable correlation between M26 and M19/M20 in Fig. 6c could be an indicator of its track.

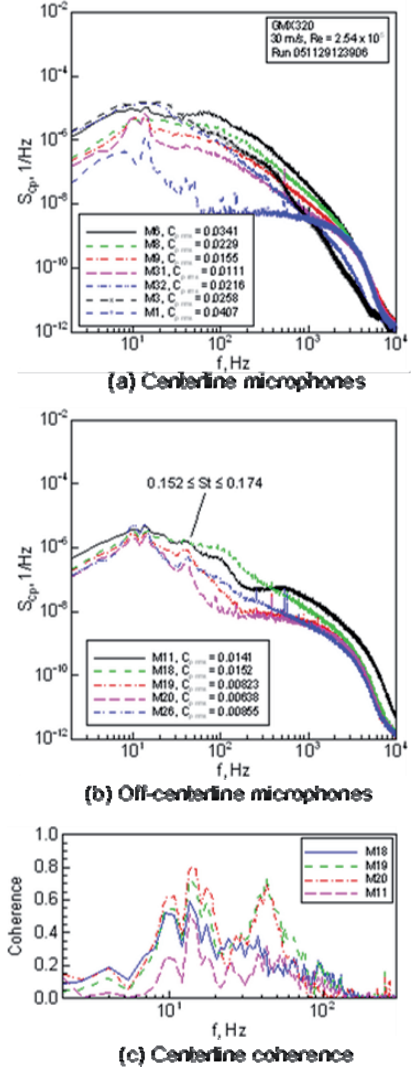


Fig. 6 The mean-square spectral distributions of C_p and coherence functions for the GMX320.

Mean Velocity Field

Typical contour plots of the mean velocity field captured by the PIV measurements in the x - z and x - y planes are shown in Figs. 7 and 8 for the GMT360 mirror, and in Figs. 9 and 10 for the GMX320 mirror. The velocity contour plots are compositions of two overlapping segments that captured an adequate length of the main-body wake in the measurement plane. It should be noted that the contour plots are not blended at the juncture of the overlapping segments, nor have the data sets of the segments been integrated. Instead, a composite was formed by overlaying the segments after transforming the local coordinate system of each segment to a common system for the measurement plane (the x_2 - y_2 and x_2 - z_2 coordinate systems) with its origin located on the reflective surface of the mirror. Each contour plot is accompanied by a series of u -component velocity profiles that exemplify the development of the streamwise velocity component within the wake of the main body. The profiles were extracted from the mean velocity field at intervals of equal distance.

In Fig. 7, a number of key features of the mean flow field can be seen in the x - z plane of the GMT360. These features are the region of local flow acceleration on the top of the mirror, the high-gradient shear layer that emanates from the top of the mirror, and the recirculation zone over the reflective surface of the mirror. Of particular note is the strong tendency for the shear layer and the freestream to be deflected towards the ground plane. The u -component profile closest to the reflective surface ($x_2 = 75$ mm) depicts a large velocity deficit and also a high-gradient shear layer at the top of the mirror. By $x_2 = 224$ mm the shear layer has diminished and the profiles farther downstream reflect a significant degree of recovery of the velocity deficit.

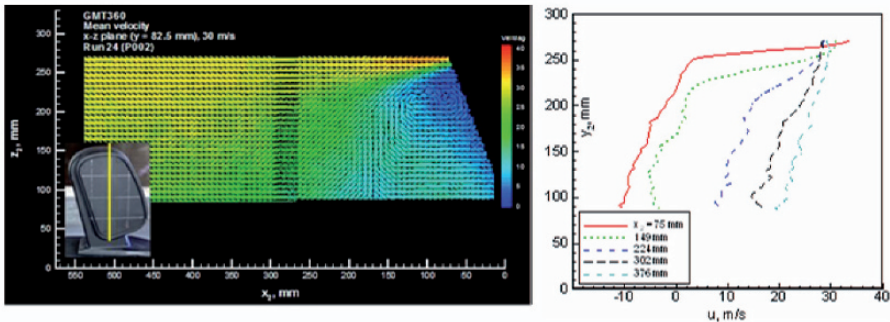


Fig. 7 A typical mean velocity field in the x - z plane and corresponding u -component profiles for the GMT360 mirror model. In the velocity field, the magnitude of mean velocity is represented by colored contours, and arrows indicate the local flow direction; the inset identifies the measurement plane.

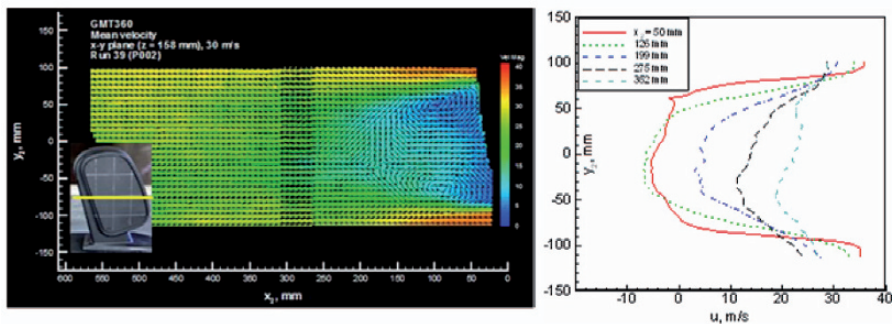


Fig. 8 A typical mean velocity field in the x - y plane and corresponding u - component profiles for the GMT360 mirror model.

In Fig. 8, the recirculation zone, as viewed in the x - y plane, features two distinct regions of near-stagnant velocity across the face of the reflective surface. Corresponding with these areas are asymmetrical time-averaged vortex structures that alludes to the presence of an alternating trailing vortex. The sequence of u -component profiles in Fig. 8 reveals a velocity recovery comparable to that seen in the x - z plane.

The PIV results for GMX320 reveal a different wake structure. In the x - z plane, Fig. 9, the shear layer at the top of the mirror is not deflected downward and appears, instead, to remain inline with the freestream flow. In addition, a vortex structure appears near the bottom of the reflective surface, and, at the bottom of the view, there is a significant degree of upward flow, which seems to originate from the region of the neck and the base. In the x - y plane, Fig. 10, the hint of a vortex structure on the border of the recirculation zone is insufficient evidence to confirm the presence of *Arch*-type vortices trailing the GMX320, as suggested earlier.

In Figs. 9 and 10, the selections of the velocity profiles taken from the wake of the GMX320 mirror are reasonably consistent, in the respective planes,

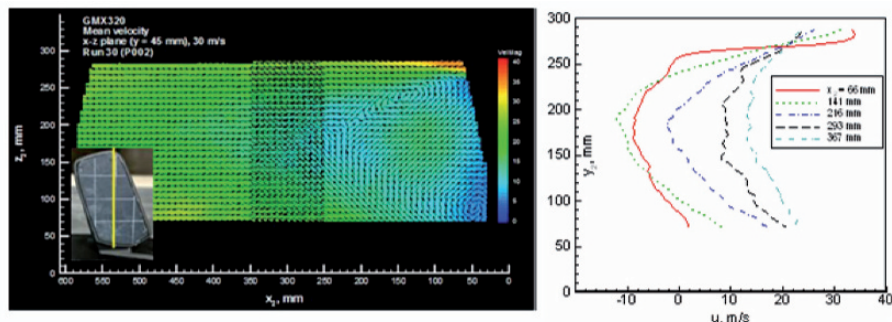


Fig. 9 A typical mean velocity field in the x - z plane and corresponding u -component profiles for the GMX320 mirror model.

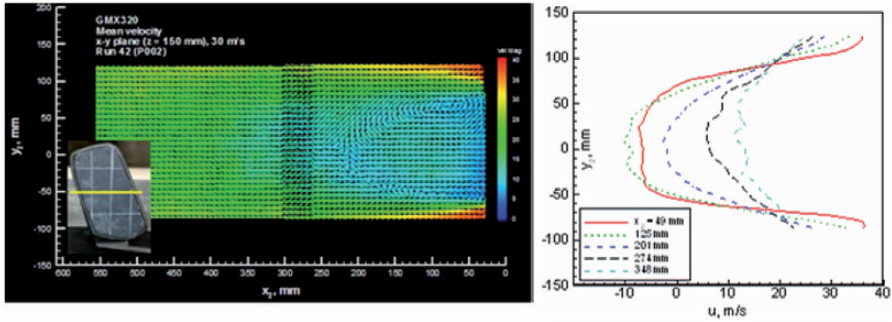


Fig. 10 A typical mean velocity field in the x - y plane and corresponding u -component profiles for the GMX320 mirror model.

beyond 125 mm from the reflective surface. When compared to the wake of GMT360, however, the recovery of the velocity deficit in the wake of GMX320 has a spatial lag. This observation can be readily confirmed by a direct comparison of the mean flow fields in the x - y plane of both mirrors, shown in Figs. 8 and 10.

4 Closing Remarks

Tests in an open-jet wind tunnel were undertaken to examine experimentally the unsteady flow fields behind automobile side-view mirrors. The tests were conducted to characterize the mean and unsteady surface pressures in the wake of two full-scale side-view mirror models, and also to describe the mean velocity field of each wake with PIV measurements. The distinctive geometric of the mirror models may be responsible for significant differences in the makeup of the respective wakes. The results of the mean pressure measurements, for instance, show that the wake of the GMT360 mirror appears to be sensitive to Reynolds number. Meanwhile, unsteady pressure measurements appear to reveal a difference in the character of the trailing vortices in the wake of each mirror. Moreover, PIV measurements depicted wake structures that are as distinctive as the geometry of the mirrors. The experimental data arising from these tests will be used to validate a computational fluid dynamics code that is a component of a larger scheme to predict aero-acoustic noise numerically.

References

- Lee I, Sung HJ (2002) Multiple-Arrayed Pressure Measurement for Investigation of the Unsteady Flow Structure of a Reattaching Shear Layer. *Journal of Fluid Mechanics* 463: 377–402
- Okuda Y, Taniike Y (1988) Flow Visualization Around a Three Dimensional Square Prism. *Journal of Wind Engineering*, No. 37, October. Presented at the International Colloquium on Bluff Body Aerodynamics and Its Applications, Kyoto

Investigation of Buoyancy Driven Flow in a Simplified Full Scale Underhood – PIV and Temperature Measurements

P. Merati¹, C.H. Leong¹, K.H. Chen², J.P. Johnson²

¹ Department of Mechanical and Aeronautical Engineering, College of Engineering and Applied Sciences, 4601 Campus Drive, Western Michigan University, Michigan, USA

² General Motors Corporation, 30200 Mound Rd., Warren, Michigan, USA
parviz.merati@wmich.edu

Abstract Results of the detailed flow measurements for an underhood buoyancy driven flow using a simplified full scale model of an engine compartment, engine block, and exhaust heaters are presented. The engine block surface temperature and exhaust heaters are kept at about 100 °C and 600 °C, respectively. This investigation reveals the complex dominant flow structures and thermal behavior of a vehicle underhood under steady state and transient states.

1 Introduction

The objective of this project was to measure the magnitudes of the flow velocity components, temperature values on the engine block and the enclosure, and air-flow temperatures under the hood. These measurements were conducted for a full scale simplified underhood configuration without openings under steady state conditions. In addition, feasibility studies were conducted to observe the thermal-flow behavior of this underhood flow for a transient condition as the system goes through its cooling cycle. The simplified underhood geometry that is used for this study is a modified version from an earlier GM R&D's underhood model [1]. The data provided can be used to validate the available and future computational codes for this complicated flow. Buoyancy convection heat transfer plays a dominant role in a vehicle underhood environment during "soaking". The "soak" condition occurs after the vehicle endures high thermal loads due to performing a sequence of operating conditions such as highway driving and trailer grade loads in hot ambient environment (>38 °C). The vehicle is then parked in a windbreak, and power is shut down. Due to the absence of any underhood airflow from the fans or the ambient surroundings, the underhood begins a thermal process that is dominated by buoyancy driven flow. During this soak process, the temperature of the engine

compartment components can rise in a tightly packed underhood before they start the cooling process.

The work published by Franchetta, et al [2] concentrated on experimental and computational investigation of a half-scaled underhood compartment using PIV, thermocouples, and commercial software VECTIS. These studies [2] were carried out for the transient natural convection condition with openings on the engine compartment model. Franchetta, et al [3] used the data obtained in [2] to validate a developed procedure within VECTIS predicting the “soak” condition with substantially reduced CPU time compared with standard fully transient CFD codes. The PIV and thermal measurement experiments carried out by Merati, et al [4] were conducted for a $\frac{1}{4}$ scale model resulting in basic understanding of the flow and thermal behavior of this buoyant flow. These experiments were carried out for a closed enclosure under steady-state condition. Three dominant 3-D laminar coherent structures were found to exist at the top of the engine compartment that agree strongly with flow visualization animations and measured air temperature: two counter rotating vortices on the top right corner and over the engine surface and one on the top left corner. The numerical results presented by Chen, et al [5] used the boundary conditions generated in [4] and computed the thermal and flow structure within the enclosure. The numerical results were in good agreement with the experimental results.

The geometry used for the current investigation consists of a glass engine compartment, trapezoidal shaped aluminum engine block and two exhaust pipes. This geometry allows for a simple experimental testing and computational modeling while retaining the general shape and main components of the engine compartment. Additionally, it is interesting to note that the geometry investigated here is a combination of several classical buoyancy driven flow experiments. They include the heated inclined and horizontal surfaces, and heated horizontal cylinders in the proximity of flat inclined surfaces.

This study is a first detailed investigation of this buoyant flow for a full scale model under steady and transient conditions. The detailed fluid flow and thermal results and the physical phenomena obtained from these measurements are valuable information for verification of the computational results and automotive engineers. Future study will concentrate on similar measurements for open enclosure under steady state and transient conditions.

2 Experimental Setup and Procedure

A simulated underhood environment was constructed for the study of buoyancy driven underhood air flow and heat transfer. The experiment was designed to give optical visibility to obtain PIV measurements of airflow and access for measurement of local air temperatures. The geometry used was kept simple, and close control of the temperature of important boundaries was made to aid in computer modeling. The underhood environment consists of the enclosure, the block, and the ex-



Fig. 1 Complete simulated underhood environment

haust heaters. The complete underhood environment is shown in Fig. 1. A scale drawing of the experimental setup with overall dimensions is shown in Fig. 2.

The enclosure consists of the base, back wall, glass cover, PIV calibration grid, and seeding mechanisms. The back wall contains patterns of holes that provide access for thermocouple measurements of air temperatures. The layout of these holes is shown in Fig. 3. The PIV calibration grid is a 0.125" sheet of aluminum cut out to fit around the block and exhaust heaters. The grid is mounted to two rods that allows it to be moved forward to the plane of interest. An image is taken of the grid before any PIV measurements are made to allow for scaling of the image and stitching of many images into a composite velocity profile. The seeding consists of injection through a small tube in the back wall.

The block, representing the engine block, oil pan, engine heads, and intake manifold, consists of a sealed chamber for holding the water that maintains the temperature of the block, and a shroud that locates and hides the thermocouples used to map temperature profiles along the block surface. A spray manifold is located inside the sealed chamber to maintain a constant film of water on all the inner walls of the sealed chamber. An immersion heater is placed inside the sealed chamber for heating of the water and block.

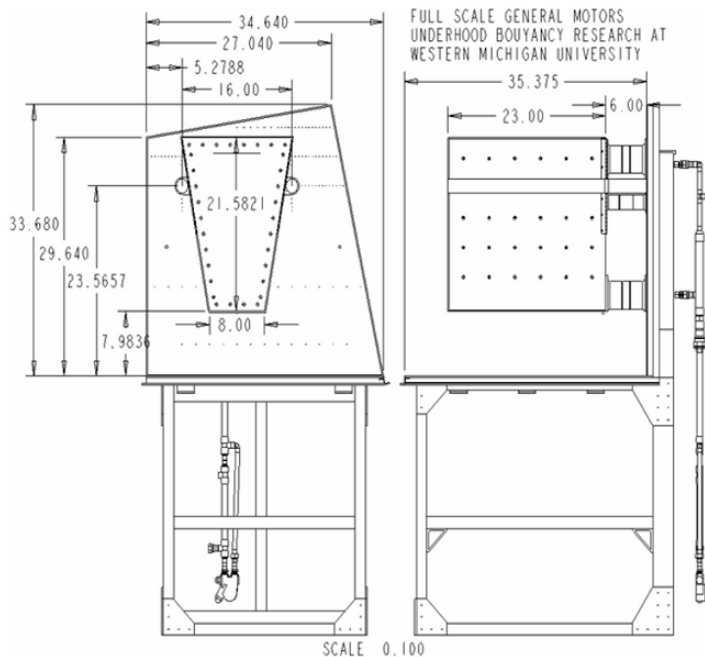


Fig. 2 Overall dimensions of the experimental underhood environment (dimensions in inches)

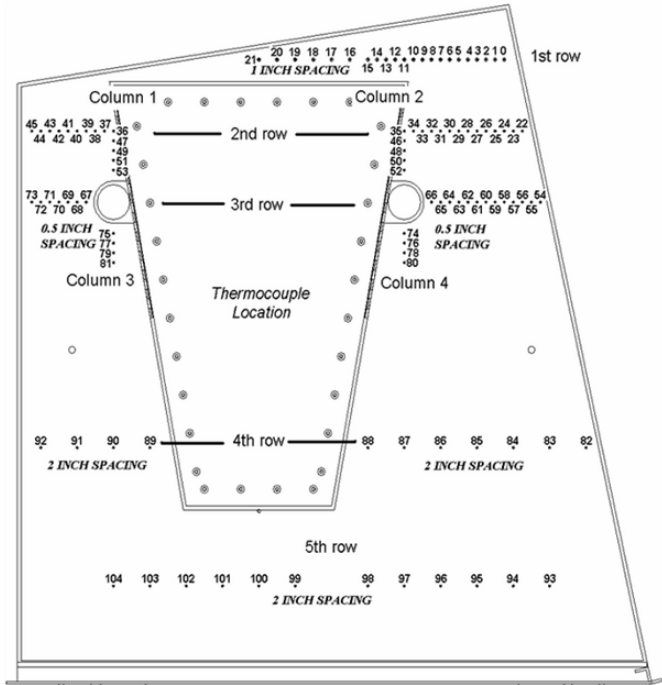


Fig. 3 Locations for air temperature measurements

The shroud was constructed of 0.250" aluminum plate on the back and 0.125" aluminum plate on the sides, top, and bottom. The shroud allows for thermocouples to be located at many points on the block without disturbing airflow, which will allow for validation of consistency from one run to the next. The thermocouples are placed in grooves machined into the shroud plates as shown in Fig. 4. The majority of thermocouples are located on the sides near the exhaust heaters as this is where the largest temperature gradients are expected.

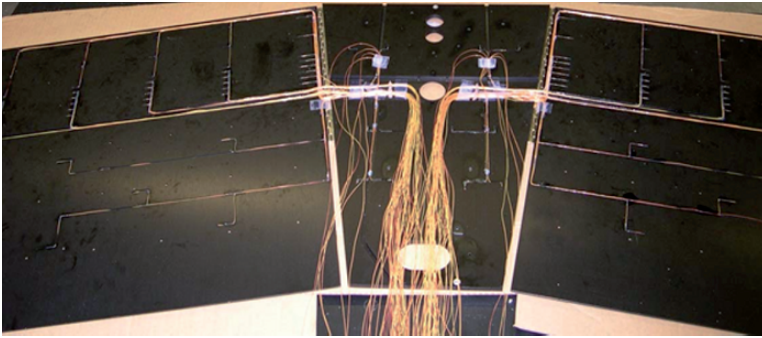


Fig. 4 Thermocouple routing in block shroud

The exhaust heaters, representing the exhaust manifold, consist of a cartridge heater, a steel shroud, a ceramic insulator, and a standoff. The control of temperature for the simulated underhood environment required a water circulation loop and a relay bank control of power. The relay bank consists of four relays that are controlled by the data acquisition system. Two are used to control the power to the two exhaust heaters, one is used to control power to the immersion heater in the sealed chamber, and the last one switches the power to the circulation pump on and off. The control of exhaust heater surface temperature relies on a feedback control loop written into the LabView software, and the surface temperature of the block is controlled through the use of the heat pipe concept. The heat pipe allows for control of surface temperature without a control loop, and the use of embedded cartridge heaters in the exhaust heaters allows for the 600 °C. The control of the exhaust heater temperature is obtained by using relays to turn the cartridge heaters on and off. Maintaining the surface temperature of the block at 100 °C is achieved by relying on the fact that for a pure fluid, the temperature of the liquid vapor interface is a function of pressure only. This is maintained by the evaporation with a slight increase in temperature, and condensation with a slight decrease in temperature. The effect is the strongest when a chamber is filled entirely with a single fluid. In this case, water is used as the working fluid. By allowing the chamber inside the block to vent to atmosphere, the evaporation of water inside the chamber displaces the air, and the action of boiling on the heater element also serves to drive dissolved gasses in the water out of solution. This allows for the chamber to be filled completely with water and water vapor after the system has been operating for a length of time. The system is then operated at saturation temperature under the atmospheric condition inside the engine block.

A reservoir of water is maintained in the bottom of the block, and water is circulated through a bank of atomizing nozzles to maintain a thin layer of water on all surfaces inside the block. This thin layer of water creates the liquid-vapor interface needed for this design to function properly. Although the reservoir of water insulates the bottom four inches of the block, the low heat transfer rate on the outside of the block allows the forced convection of the water to remove or add heat as necessary.

Thermal Measurements

Each engine block surface, except front surface, is embedded with thermocouples. External thermocouples are attached to the engine block front surface and enclosure surfaces. The temperature measurements are taken by four-32 channel temperature data conditioning modules and a National Instrument data acquisition card. Furthermore, these measurements are monitored and recorded by a LabView code developed at WMU. The results are then converted to the Tecplot format (FE Block, cell-centered). The thermocouple mesh size and associated areas for each of the engine block and enclosure surfaces are shown in Table 1. The temperature values are obtained at the center of the cells. The temperature mesh distribution and chosen coordinate system located at the point where the inner surfaces of the vertical, bottom, and back enclosure walls intersect are shown in Fig. 5. The temperature of the twelve thermocouples embedded in the two exhaust heater shrouds and the three thermocouples in the water circulation loop are monitored to assure that the system has reached steady state condition.

Table 1 Thermocouple grid

	Location, Side (s)	Thermocouple Grid	Average TC Cell Area (cm ²)
Engine Block	Front (I)	4 × 9	48.1
	Back (II)	4 × 3	150.3
	Right (III)	5 × 11	82.9
	Left (IV)	5 × 11	82.9
	Top (V)	2 × 3	390.3
	Bottom (VI)	2 × 3	196.9
Glass Enclosure	Front (VII)	7 × 9	102.7
	Right (VIII)	9 × 11	123.6
	Left (IX)	9 × 11	107.3
	Top (X)	9 × 11	99.4
	Back (XI)	7 × 9	102.7
	Bottom (XII)	4 × 7	283.3

The engine block is heated internally by water near saturation and externally by the exhaust heaters. The enclosure and the space within the enclosure are heated by the engine block and exhaust heaters. The exhaust heaters are heated to 600 °C, and water spray temperature inside the block is kept at 95 °C. The exhaust heaters, water temperature inside the block, and the average block surface temperature reach their desired steady state temperature after 3,600 sec. All tests are then conducted at this steady state condition except for transient experiments where the system is allowed to reach the steady state and then all heating is turned off for the cool down period.

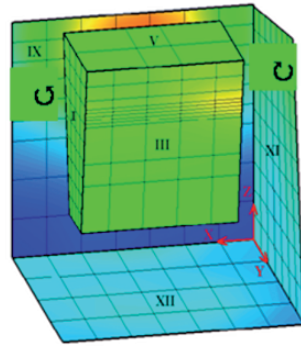


Fig. 5 The chosen coordinate system

Particle Image Velocimetry

The simulated engine compartment and measurement setup for 2D PIV measurements are shown in Fig. 6. Paths A and B are the two orientations used to measure the flow velocities on the left and right sides of the engine block. PIV is conducted on three vertical front, middle and back planes corresponding to $X = 317$ mm, 444 mm, and 571 mm, respectively. This configuration allowed most of the regions on the sectioned planes to be captured. The exception was the small gap between the cylinder and the engine block side wall where the CCD camera sensor saturated from the reflections on the neighboring surfaces, labeled shadow regions in Fig. 6.

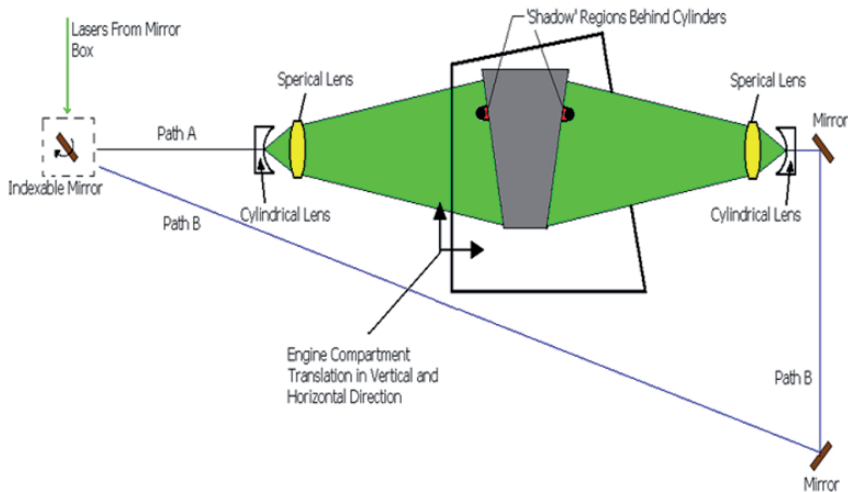


Fig. 6 The simulated engine compartment and 2D PIV setup

The camera field of view for these PIV tests was 12 cm by 12 cm in size. An interrogation area size of 32 X 32 pixels corresponding to physical size of approximately 4 mm X 4 mm provided adequate signal to noise ratio for repeatable results. Thus, to capture velocity field one region at a time, each vertical plane was broken up into 49 separate regions. With this limited field of view, the measurement regions are overlapped to increase vector density and avoid discontinuities near the regions' borders.

Particle Seeding

For seeding, chalk particles are used since they can sustain the high temperature environment around the hot exhaust heaters unlike the commonly used olive oil aerosol. Chalk particles are adequately small in size and weight to capture the flow details within this mostly low speed buoyant flow. A new fluidized bed solid aerosol generator was developed for PIV seeding. Using this seeder, chalk particles are injected into the engine compartment from the back plate holes once the system reaches steady state conditions. Flow disturbances due to particle injection damps out approximately 45 seconds after seeding as observed on the PIV camera. Thus, after 45 seconds, PIV recording sequence was started to capture 200 image pairs at 15 Hz.

3 Results

Engine Block Surface Temperatures

The temperature contours for the top (V), bottom (VI), right (III), left (IV), back (II), and front (I) faces of the engine block are presented in Fig. 7 and the corresponding statistics are presented in Table 2.

The discontinuity of the temperature magnitudes at some of the corners, especially between the right (III) and back (II) faces is due to the large mesh size for the temperature measurements.

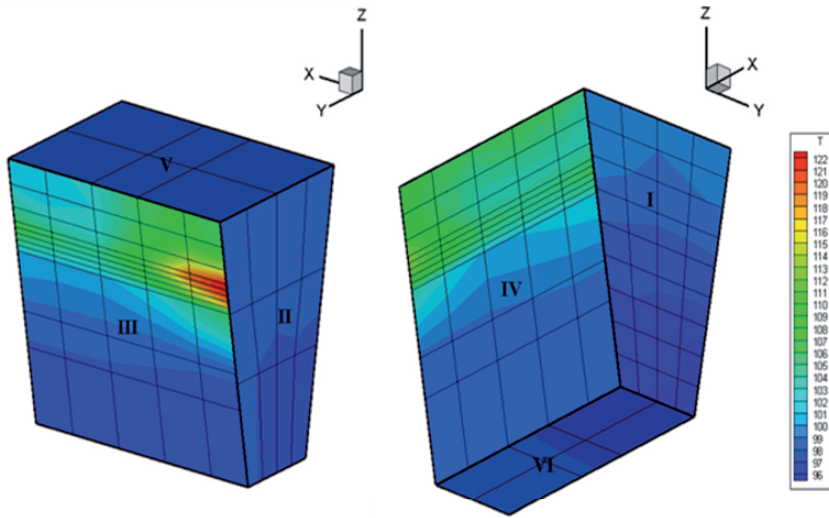


Fig.7 Engine block surface temperature contours

Table 2 Engine block surface temperature statistics

Engine Block Surface Temperature C°						
	Front (I)	Back (II)	Right (III)	Left (IV)	Top (V)	Bottom (VI)
Mean	97.6	98.2	105.1	103.8	98.6	97.9
Std Dev	1.4	0.5	7.0	4.2	0.1	0.5
Max	99.7	98.9	123.2	113.6	98.8	98.2
Min	94.3	97.2	97.6	98.2	98.4	97.1
Range	5.3	1.7	25.6	15.4	0.4	1.2

The thermal contours presented assume constant temperature over a cell area and does not take into account its thermal gradient. This is because temperatures were only obtained at the center of cells. Measurement of thermal gradients over the cells requires extensive time and effort.

There are small temperature gradients at the front (I), back (II), top (V), and bottom (VI) surfaces. The temperature is slightly higher at the top corners and becomes lower as the distance of Z-axis decreases for the front face. The highest temperature value is found at the right (III) surface closest to the exhaust heater. At the right (III) and left (IV) surfaces, the temperature is high along the axis of the exhaust heaters and becomes lower as the distance from the heaters increases. In addition, the temperature is lower at the bottom of each surface. Temperature gradients exist along the X-axis at the right (III) and left (IV) surfaces.

In the upper back corner of the left and right faces, the temperature is high near the back face, more noticeable on the right (III) surface. In order to verify that this is a real effect and not due to the design of the engine heating system, the engine block was heated to the desired temperature without turning on the heaters for the exhausts. Under this condition, a uniform surface temperature for the engine block is expected. Any deviation from this is most likely due to the errors caused by the thermocouple measurements and the engine block heating system. The surface temperature gradients are found to be much smaller than what were observed with the exhaust heaters at 600 °C. Thus, the observed thermal behavior of the engine block surfaces is caused by the radiated and convected heat from the exhausts and not due to the experimental errors.

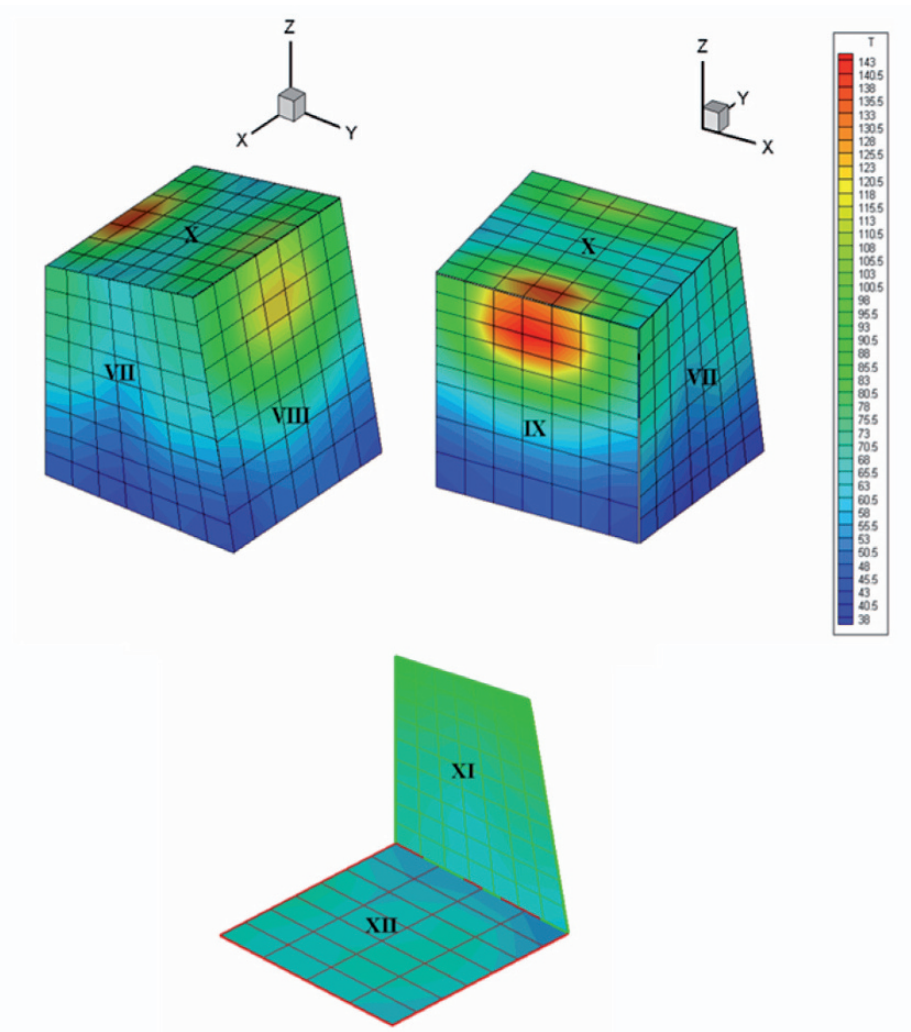


Fig. 8 Enclosure surface temperature contours

Based on the computational results for a $\frac{1}{4}$ scale geometry by Leong⁷ (engine block surface temperature of 120 °C, enclosure wall temperature of 100 °C, and exhaust surface temperature of 600 °C), two large secondary flows shown schematically in Fig. 5 were observed. If the existence of these flow structures could be validated by future experimental work, the larger temperature regions observed on the engine block side surfaces toward the end faces are justified. The vortex structures squeezed between the end faces of the engine block and the front and back sides of the enclosure are the main cause of three dimensionality of the flow near the end faces of the engine block. Thus, the heat of the exhausts at these locations are not mostly carried away toward the top face of the enclosure wall similar to the middle region of the exhaust (reason for higher temperatures observed near the midplane of the enclosure walls, Fig. 8). Instead, near the ends, heat of the exhaust and thermal boundary layers on the engine block surface is constantly convected along the exhaust axis by the end region secondary flows. This effect generates higher temperature on the engine block surface and lower temperature on the enclosure walls near the front and back sides of the engine block. There are less thermal gradients on the enclosure walls on the right side relative to the left side due to existence of turbulent flow observed over the plume near the top regions of the enclosure walls on the right half of the enclosure.

Enclosure Surface Temperatures

The temperature magnitudes are measured on the outside surfaces to prevent the thermocouple wires from disturbing the flow inside the enclosure. Because of the design constraints, the temperatures on the back (XI) and bottom (XII) faces had to be measured on the inside. The enclosure surface temperature contours and statistics are shown in Fig. 8, Tables 3 and 4, respectively.

The high temperature regions are located directly above the exhaust heaters on the top face and closest to the exhaust heaters on the right (VIII) and left (IX) faces.

Table 3 Enclosure inner surface temperature statistics

	Enclosure Inner Surface Temperature °C	
	Back (XI)	Bottom (XII)
Mean	72.9	59.7
Std Dev	7.3	10.4
Max	89.3	77.7
Min	61.3	40.4
Range	28.1	37.3

The high temperature of these regions is mostly caused by radiation from the exhaust heaters as described in the following section. Temperature distributions on the front surface (VII) show similar characteristics as the engine block front surface. The temperature is highest at the top two corners with thermal gradients along the Y and Z axes. The back face is warmer than the bottom surface with an obvious rise in temperature approaching the exhaust heaters.

Table 4 Enclosure outer surface temperature statistics

	Glass Outer Surface Temperature °C			
	Front (VII)	Right (VIII)	Left (IX)	Top (X)
Mean	59.7	80.6	84.4	83.8
Std Dev	10.4	24.7	32.6	24.0
Max	77.7	124.4	147.2	146.4
Min	40.4	41.2	39.1	58.8
Range	37.3	83.2	108.0	87.6

Temperature Difference across the Wall Thickness of the Enclosure at the Middle Plane

The outside and inside wall temperatures for the right (VIII), top (X), and left (IX) surfaces are measured at the middle plane of the enclosure. The inside and outside wall temperatures for the enclosure wall are shown in Fig. 9. ΔT is the temperature difference across the wall thickness of 3/16", and r is the distance of the outer thermocouple location from the center of the exhaust heaters

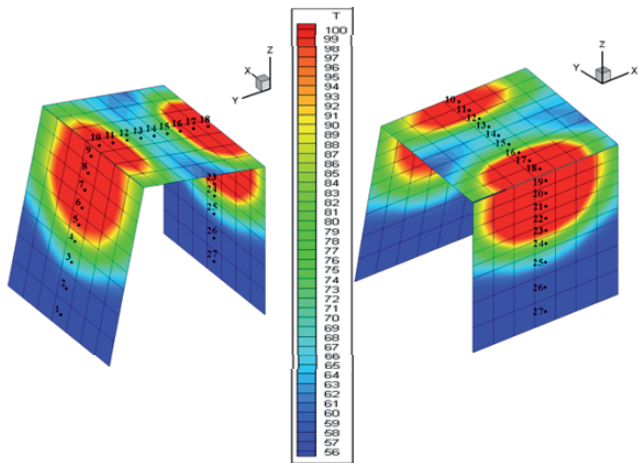


Fig. 9 Middle plane thermocouple locations

As the magnitude of the local temperature becomes higher, ΔT increases, representing larger heat flux across the enclosure wall. The largest temperature difference is observed at thermocouple 17 which is located directly above the left exhaust heater. In addition, on the side walls (VIII and IX), the highest temperature magnitudes occur at thermocouples 7 and 20 with the smallest distance from the neighboring exhaust heaters. Since the highest temperatures occur where the enclosure walls are exposed to the exhaust heaters, it is suspected that most of heat in the hottest regions of the enclosure wall is transferred through radiation. Since heat flux due to radiation is proportional to $1/r^2$, for the hottest regions, the dominance of radiation heat transfer is validated by the correct predicted trend of the $1/r^2$ curves in Fig. 10. On a section of the top surface, the $1/r^2$ curves are not plotted because view of the exhaust heaters is blocked by the engine block. In this region, the radiation is assumed to be significantly smaller consistent with the observed lower temperatures.

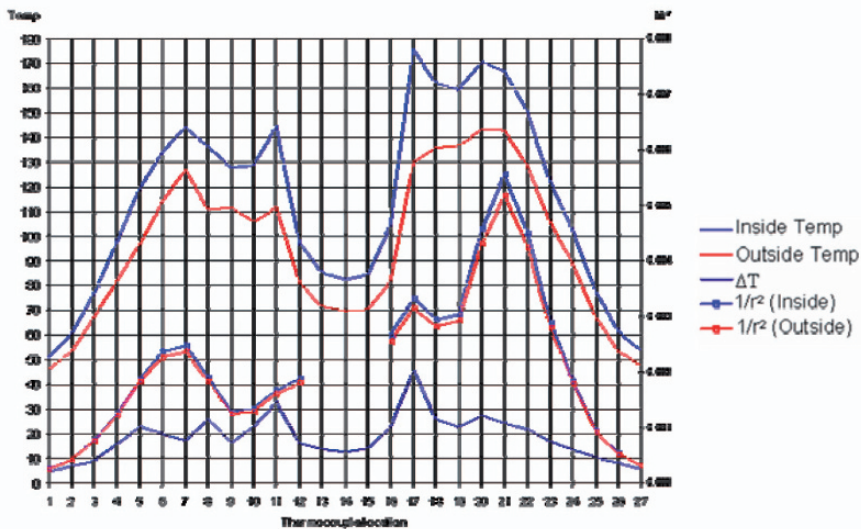


Fig. 10 Inside and outside wall temperatures for the enclosure wall at the middle plane

Air Temperature

Air temperature measurements are obtained on the middle plane with the location labeled in Fig. 3. Since active flow structures in the quarter scale are concentrated in the upper portion of the enclosure, most of the temperature measurements were conducted here. Due to structural integrity of the back plate as part of the support for the engine block, extensive access holes were not implemented. Response characteristics of the thermocouple used for air temperature measurements showed that a 5 seconds cut off and a 45 seconds recording time for actual air temperature

are acceptable. The data shows a general slow increase in temperature since the thermocouple is exposed to not only hot fluid but also radiation from the heated cylinders. Therefore, a cut off time is required.

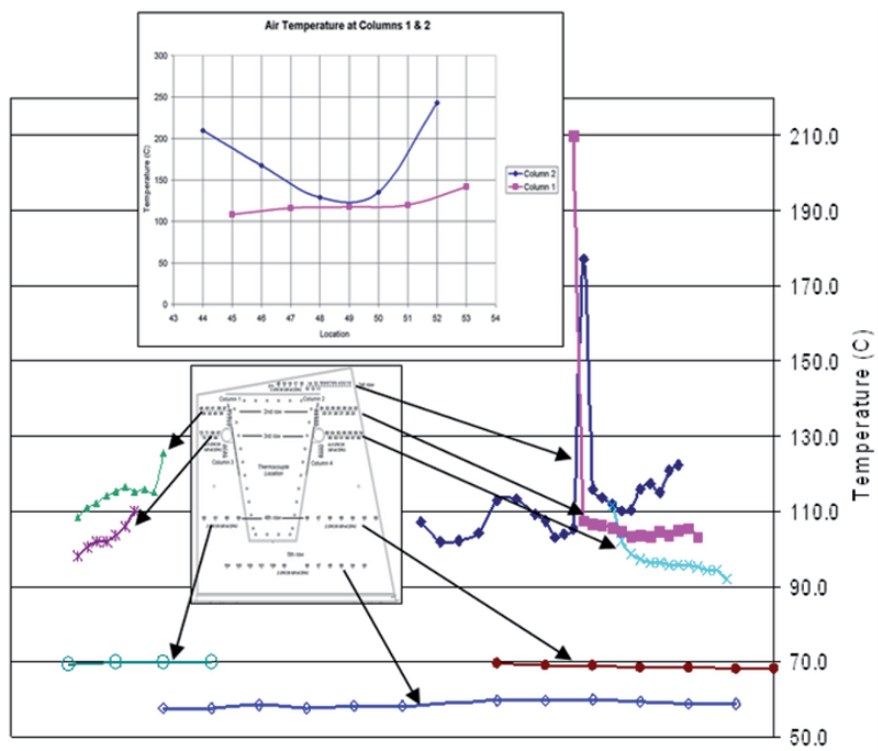


Fig. 11 Air temperature profiles

Figure 11 shows the results of the air temperature measurements using unshielded thermocouples starting from top row. Air temperature on the left side is consistently higher than the right side. The temperature profile for the first row shows the presence of the hot plume and the lobes on the side are consistent with the mixing due to the secondary flows observed by the PIV measurements. Air temperatures of the column 2 thermocouples presented in Fig. 11 are higher than column 1. This is an indication of a hotter thermal layer around the right exhaust heater and the upper right engine block surface. This can only be verified with extensive thermal boundary layer measurements with accurate traversing thermocouple measurements to resolve the large temperature gradients in these regions. The bottom regions, rows 4 and 5 are much colder than the top regions as expected.

Velocity Measurements

Contours of velocity magnitude and standard deviations and their close up views for vertical middle plane are shown in Figs. 12, 13, and 14, respectively.

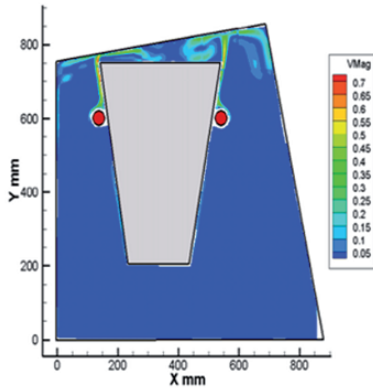


Fig. 12 Contours of velocity magnitudes

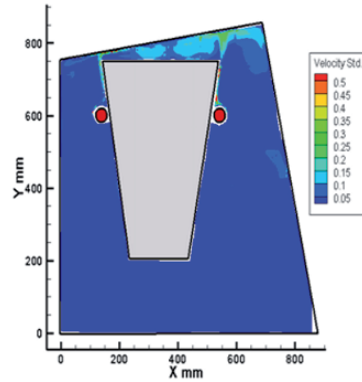


Fig. 13 Contours of velocity standard deviations

Visualized flow with stream traces is shown in Fig. 15. Most of the flow activity within this buoyant flow occurs near the top of the compartment above the exhaust heaters and in the region between the top of the engine surface and the top enclosure wall. The maximum measured velocity magnitude is about 0.72 m/sec above the exhaust heaters. Large flow standard deviations are seen over the engine block and the right top side of the compartment. Two large counter-rotating vortices are observed in Fig. 14.

The flow within this region is turbulent as demonstrated by animation movie created from the instantaneous velocity vector fields above the right exhaust heater. Since the distance from the top of the exhaust heater from the top enclosure wall is larger for the right side compared with the left side, the hot plume on the right has more time to go through flow transition and turbulent stages. The turbulent jet finally impinges on the top surface of the enclosure wall. This phenomenon is clearly shown by six instantaneous realizations of the flow structures in Fig. 16 as the plume cycles through one period of overall flow oscillation. Since PIV was conducted with the laser pulse frequency of 15 Hz, the interval between consecutive images is 0.0667 sec. With six images contributing to a full cycle, the overall frequency of this sweeping turbulent impinging jet is about 2.5 Hz. The entrainment due to the plume shear layer is clearly shown by the ensemble average of velocity traces above the right exhaust heater in Fig. 17.

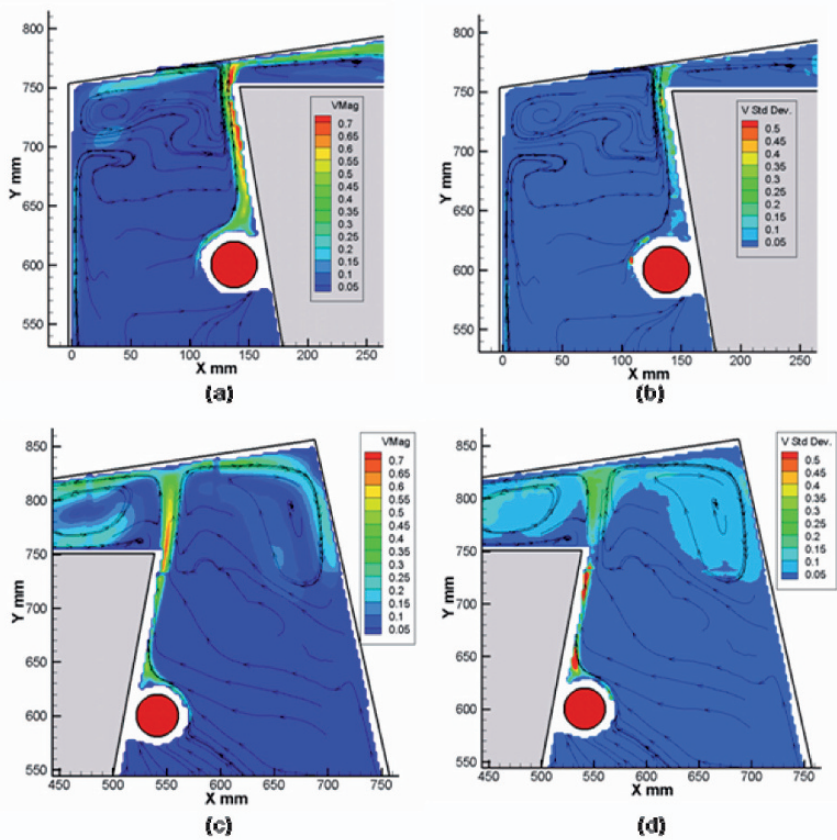


Fig. 14 Regional close up display of velocity contours with stream tracers

The temperature profiles on the enclosure inner surface at middle plane (Fig. 18) show that the wall temperatures are hotter on the left side. In addition, due to the turbulent flow of the top right side, thermal gradients are smaller on the top right side of the enclosure wall relative to the left side

The onset of instability waves and transition to turbulence is clearly observed by visualizing a long candle flame behavior

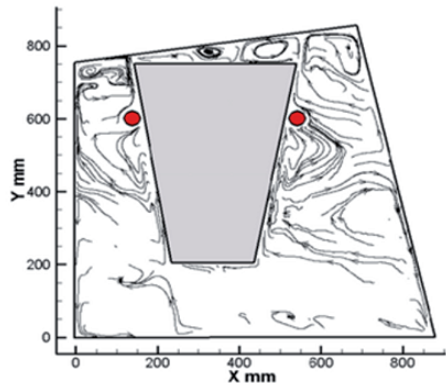


Fig. 15 Stream traces on middle plane

Although flame propagation is not exactly the same as the buoyant flow around a heated cylinder, the hot flame and smoke downstream of the candle flame shows a phenomenon very similar to the hot plume of the exhaust in this experiment. These findings are also consistent with the results of tests performed by T. Maxworthy⁶ who generated a hot thermal plume from propane diffusion flames and observed oscillations similar to the ones that are observed here. Extensive studies have been made on forced and cross flow convection of buoyancy driven flow over a cylinder at very low Reynolds numbers. Unfortunately, the authors have not found any work on pure natural convection buoyant flow over a cylinder.

The critical Grashof numbers for flow around a vertical flat plate and around a cylinder are about 10^9 . The Grashof number is the ratio of buoyancy force to viscous force as defined by the following equation

$$Gr_{L_c} = \frac{g\beta (T_s - T_\infty) L_c^3}{\nu^2} \quad (1)$$

where $\beta = 1/T$ is the volume expansion coefficient
 T_s is the surface temperature of the object
 T_∞ is the fluid temperature far from the surface
 L_c is a characteristic length of the geometry and
 ν is the kinematics viscosity of the fluid at $T_f = (T_s + T_\infty)/2$

Calculated Grashof numbers for different geometrical shapes that make this full scale model result in values less than the critical value. However, downstream of the right heated exhaust, flow transitions from laminar to turbulent flow. Thus this flow is composed of different flow regimes, laminar, transition, and turbulent flows depending on the flow location relative to the underhood components.

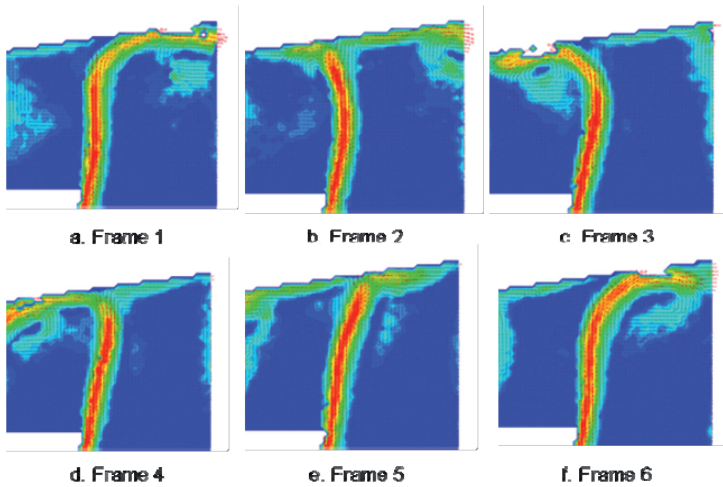


Fig. 16 Full cycle of instantaneous unsteady thermal plume on top of the right exhaust heater

Although in the lower regions of the enclosure, velocity magnitudes are in the range of 0.0001 to 0.02 m/sec, stream traces show the details of the flow. Buoyancy generated as the result of hot surfaces of the inclined engine block are the cause of higher speeds in the proximity of these surfaces. The shape of the stream traces near the inclined engine block surfaces indicates the presence of thermal boundary layers caused by hot engine block surfaces observed by a PIV animation video created for the region below the right exhaust heater. At $y \sim 400$ mm, flow diverts toward the hotter sections of the enclosure surfaces and then it is drawn toward the hotter exhaust heater. This is due to the low pressure region generated around the exhaust as a result of the buoyancy. Plots similar to those of the middle plane are obtained for the front and back vertical planes. Discussion of the flow behavior on these planes is not presented here since flow on these planes is similar in nature to the flow of the middle vertical plane described in previous section.

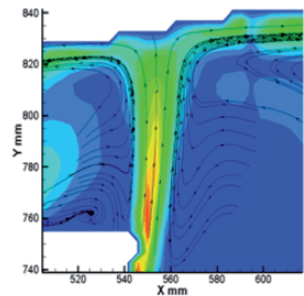


Fig. 17 Velocity traces above the right exhaust heater

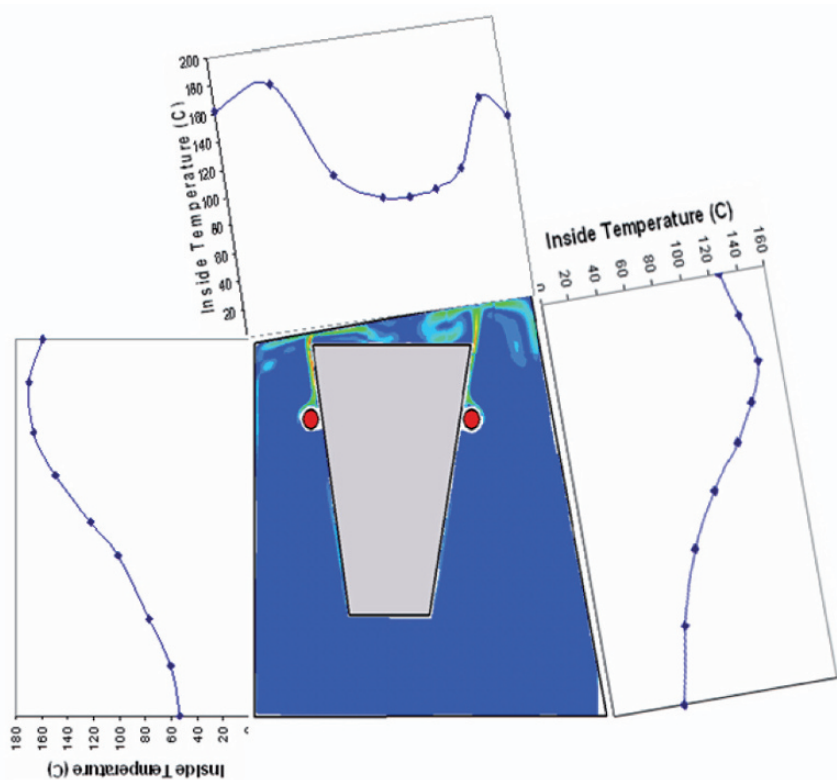


Fig. 18 Enclosure inner surface temperature profiles at the middle plane

Transient Velocity and Temperature Measurements-Feasibility Study

Final part of this project was to assess the feasibility of conducting transient PIV and temperature measurements. The challenge for PIV measurements under transient conditions is that seeding particles at very low speeds drop out of the flow field and no longer follow the flow field. At high speeds, the particles attach to the surfaces of the enclosure and the engine block due to moisture and surface roughness. As a result, seeding density has a dramatic effect on data density. However, by initial heavy seeding of the flow, high quality measurements for up to 45 minutes in the region above the right exhaust heater could be obtained without reseed-ing. PIV measurements were taken every 5 minutes at 15 Hz, recording 200 frames for averaging. This region is chosen since it is over the plume and is expected to be directly affected by thermal changes during transient period of cooling down. Flow in the rest of the enclosure is also changing during transition. However, for transient capture of the flow behavior, PIV is only possible for one region at a time due to large field of view of the entire engine compartment. The sequence of velocity magnitude contour plots in Fig. 19 displays gradual change in the flow characteristics and velocity magnitude. As the hot plume weakens, its centerline is tilted toward the right corner of the enclosure. The small vortex on top of the right corner of the engine block starts to disappear and the large vortex on the left side of the plume migrates toward the right side accompanying the plume. This overall migration might be due to a clockwise mean velocity field throughout the engine compartment due to the slant angles on the top and right surfaces. The animated videos obtained from the velocity fields show that the large oscillation of hot plume begins to die down and eventually disappear as the system cools down. Disappearing of the large oscillatory motion observed under steady state condition can be seen from decreasing magnitudes in velocity standard deviation. Although, low frequency oscillations with small amplitudes on the hot plume still exists, at about 15 minutes from the start of the transition period, large flow fluctuations are completely vanished.

Air temperature at a point in this region was measured, and block temperature was recorded throughout the process. Air temperature was measured at 2.5 minutes interval and block temperature data were obtained every 10 seconds. The data show the temperature change during transient cooling up to 45 minutes. Once all power is shut off, temperature of the exhaust heaters drops and the heat transferred to the engine block is no longer cooled to saturation by the jet sprays inside the engine block. Therefore, the right and left side of the engine block surface temperatures spike immediately after power shut down.

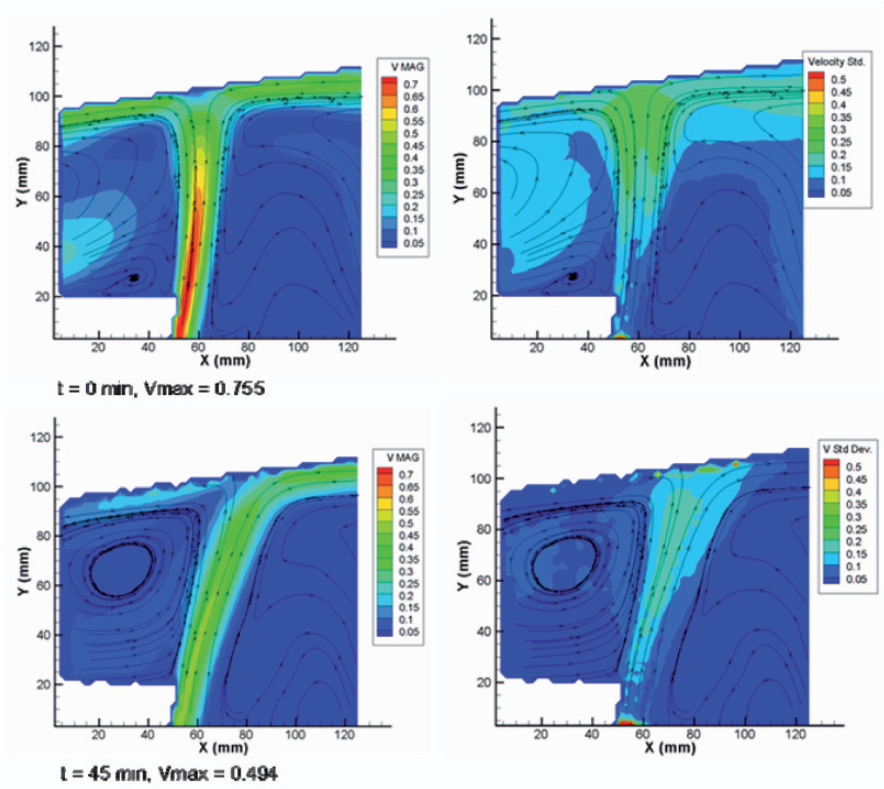


Fig. 19 Ensemble average of velocity fields for the region above the right exhaust heater under transient cooling down process

4 Conclusions

- We have found some differences between the flows for $\frac{1}{4}$ and the full scale models. This is because the driving force in this flow is the buoyancy that is created by the temperature difference between the hot surfaces and the surrounding flow. The important non-dimensionalized number is Grashof number, a measure of the relative magnitudes of the buoyancy force and the opposing viscous force acting on the fluid. Under the same thermal boundary conditions, the full scale model's Grashof number is 64 times larger than the $\frac{1}{4}$ scale model. The dominant difference between the two model flows is the transition from laminar to turbulent flow over the right cylinder which was not observed in the $\frac{1}{4}$ scale model

- For steady state condition, PIV measurements on three vertical planes at $X = 317, 444,$ and 571 mm have shown that flow is turbulent on the right top portion of the enclosure contributing to enhanced thermal convection relative to the top left side. Radiation from the exhaust heaters plays a dominant role in raising the temperature of the enclosure walls in the vicinity of the exhaust heaters.
- The measured dominant structures are two larger counter rotating vortices over the top right side of the engine block and two counter rotating vortices over the top left side. In addition, there exist two dominant smaller secondary flows, one over the top surface of the engine block near its right corner, and one in the top right corner of the enclosure.
- Entrainment of the flow contributes to thick thermal boundary layers on the exhaust heater and slant surfaces of the engine block. Three dimensional computational results conducted for $\frac{1}{4}$ scale model exhibit secondary flows between the end faces of the engine block and the enclosure walls. These secondary flows are the main cause of the three dimensionality of the flow near the end faces of the engine block. The heat of the exhausts at these locations is not mostly carried upward toward the top face of the enclosure wall similar to what is observed in the middle region of the heater. Instead, near the ends, heat of the exhaust and thermal boundary layers on the engine block surface is constantly convected along the exhaust axis by the end region secondary flows. This effect generates higher temperatures on the engine block surface and lower temperatures near the top regions of the front and back sides of the enclosure walls.
- Air temperature results under steady state condition show that generally the right side is cooler than the left. The temperature profile for the first row show the presence of the hot plume and the lobes on the side are consistent with the mixing occurring due to the secondary flows observed by the PIV measurements. Large temperature gradients over the narrow regions of the plume require a more accurate traversing mechanism for temperature measurements with higher resolution.
- PIV and thermal measurements were conducted during the first 45 minutes after the power to the entire system was turned off to investigate the feasibility of the transient cooling. PIV was only obtained for the region of the enclosure above the right exhaust heater in the middle plane. Air temperature was also measured at a point in this region. However, the entire engine block surface temperature and the exhaust heater temperatures were measured. Once transient cooling starts, the hot plume weakens and its centerline is tilted toward the right corner of the enclosure. The small vortex on top of the right corner of the engine block disappears and the large vortex on the left side of the plume migrates toward the right side accompanying the plume. The turbulent flow observed under steady state condition weakens drastically and the intensity of the

flow structures and in general flow activity begins to slow down with time. The temperature of the block surfaces start to rise for the first 7.5 minutes and then the cooling process begins. This effect is because the heat pipe mechanism of the engine block ceases to function and thus for the first few minutes, the heat of the exhausts is not removed properly.

References

- [1] *GMR report, General Motors Research and Development* 1990.
- [2] Franchetta M, Suen K, Williams P and Bancroft T, Investigation into Natural Convection in an Underhood Model Under Heat Soak Condition, *SAE Technical Paper Series*, 2005, 2005-01-1384.
- [3] Franchetta M, Bancroft T, Suen K, Fast Transient Simulation of Vehicle Underhood in Heat Soak, *SAE Technical Paper*, 2006, 2006-01-1606.
- [4] Merati P, Cooper N, Leong C, Chen K and Johnson J, Investigation of the Buoyancy Driven Flow in a Simplified Underhood-Part I, PIV and Temperature Measurements, *SAE Technical Paper Series*, 2006, 2006-01-1698.
- [5] Chen K, Johnson J, Merati P, Cooper N and Leong C, Investigation of the Buoyancy Driven Flow in a Simplified Underhood-Part II, Numerical Study, *SAE Technical Paper Series*, 2006, 2006-01-1607.
- [6] Maxworthy T, The Flickering Candle: Transition to a Global Oscillation in Thermal Plume, *J. Fluid Mech.*, 1999, Vol. 390, pp. 297-323.
- [7] Leong C, Computational Analysis of Heat Convection Buoyancy Pattern using Fluent Simulation Software, *WMU-ME 595 Project Report*, June 26, 2006.

Investigation of Buoyancy Driven Flow in a Simplified Full-Scale Underhood – Numerical Study

K.H. Chen¹, J.P. Johnson¹, P. Merati² and C.H. Leong²

¹ General Motors Corporation, 30200 Mound Rd., Warren, Michigan, USA

² Department of Mechanical and Aeronautical Engineering, College of Engineering and Applied Sciences, 4601 Campus Drive, Western Michigan University, Michigan, USA
kuo-huey.chen@gm.com

Abstract Transient numerical results are presented for simulating buoyancy driven flow in a simplified full-scale underhood in automobile. The flow condition is set up in such a way that it mimics the underhood soak condition, when the vehicle is parked in a windbreak with power shut-down, after enduring high thermal loads due to performing a sequence of operating conditions, such as highway driving and trailer grade loads in a hot ambient environment. For validation, several experiments were conducted to measure the temperature and velocity fields. The simplified underhood geometry, although simplified, consists of the essential components in a typical automobile underhood undergoing the buoyancy driven flow condition. It includes an enclosure, an engine block and two exhaust cylinders mounted along the sides of the engine block. The calculated temperature and velocity were compared with the measured data at different locations near and away from the hot exhaust plumes. The numerical predictions reveal a complex, transient flow structure under the buoyancy condition, and the results show very favorable comparisons with the experimental velocity and temperature measurements. The transient numerical procedure developed for the current study would pave the way for the future numerical simulations of the practical real underhood soaking.

1 Introduction

Buoyancy convection heat transfer plays a dominant role in a vehicle underhood environment during “soaking”. The underhood thermal soaking, or “soak”, condition occurs after the vehicle endures high thermal loads due to performing a sequence of operating conditions such as highway driving and trailer grade loads in a hot ambient environment (>38 degrees C). The vehicle is then parked in a

windbreak, and the power is shut down. Due to the absence of any underhood air-flow from the fans or the ambient surroundings, the underhood begins a thermal process that is dominated by buoyancy driven flow. During the soaking, the temperature can rise even higher in a tightly packed underhood environment.

To numerically simulate the underhood soak condition in a full size vehicle, required is a significant amount of computer resources and computational techniques. The soak process is inherently transient and involves a small scale flow motion (usually less than 1 m/s) in a very complex 3D underhood layout. Before this thermal process can be confidently analyzed in a production analysis environment, it is necessary to evaluate the numerical approaches step by step using an idealized geometry that captures the essential features of the flow physics but avoids undue complexity.

A previous experimental and numerical study of similar flow configuration with quarter-scale was conducted [1] [2]. Several numerical issues, including mesh type and size, spatial and temporal accuracy have been discussed [2]. For the quarter-scale case, both experimental measurement and numerical simulation exhibit steady state flow pattern [1] [2] and the agreement between the prediction and the experiment is very favorable in both temperature and velocity. As mentioned previously, the current study using the simplified model is to develop a viable numerical procedure to predict such buoyancy driven flow in an automobile underhood undergoing a soak condition. This development was designed to capture all the essential flow physics with the help of the experiment without involving details of the complication in the real underhood geometries. With this in mind, the quarter-scale study in [1] and [2] might not provide the correct flow physics due to size difference. Nonetheless, the experiences learned, both in simulation and experiment, from the quarter-scale study are very helpful in providing directions in devising a more sophisticated full-scale study. The full-scale configuration is almost identical to the quarter-scale. However, due to the significant increase in size, a new Particle Image Velocimetry (PIV) measuring technique, new type of seeding for the laser and lots more thermocouples for temperature fields are required. The full-scale simplified underhood geometry consists of an enclosure, an engine block and two exhaust cylinders mounted along the sides of the engine block (Fig. 1). The flow condition is set up in such a way that it mimics the soak condition in the underhood. The geometry size difference for the current full-scale study produces quite different flow and temperature fields than the quarter-scale does with a similar temperature boundary condition. Among others, the most significant difference is that both experimental and numerical predictions for the full-scale case reveal a complex, transient flow structure which does not exist for the quarter-scale case. This indicates that the thermal plume driven by the strong buoyancy force above the cylinder becomes unstable at the full-scale configuration. For the full-scale test, in addition to the constant boundary temperature condition measurement, a 45-minute transient cool-down temperature boundary condition and a limited number of temperature and velocity data above the hot thermal plume were taken during the cool-down period.

This paper is organized as follows. A brief overview of the experimental setup and measurement is presented first. Following that, the numerical study using Fluent [3] and the comparison with the measurements are discussed. The plan for further investigation is laid out as well.

2 Experimental Setup

The details of the experimental work were documented well in [4, 5]. In the current paper, the experimental data and experimental setup will be described briefly so that the numerical study can be easily explained. The flow configuration is shown in Fig. 1 in which depicted an outer enclosure, an engine block and two exhaust cylinders mounted along the sides of the engine block. The size of the outer enclosure in the simplified underhood shown in Fig. 1 is roughly 1m x 1m x 1m and it has a slanted hood and sloped dash which mimic the underhood compartment in a production vehicle. The two exhaust cylinders are identical in length and size and they are 584.2 mm long and 45.7 mm in diameter. A brief sketch with dimensions for the experimental setup is shown in Fig. 2. The detailed geometry with dimensions is documented in [4, 5].

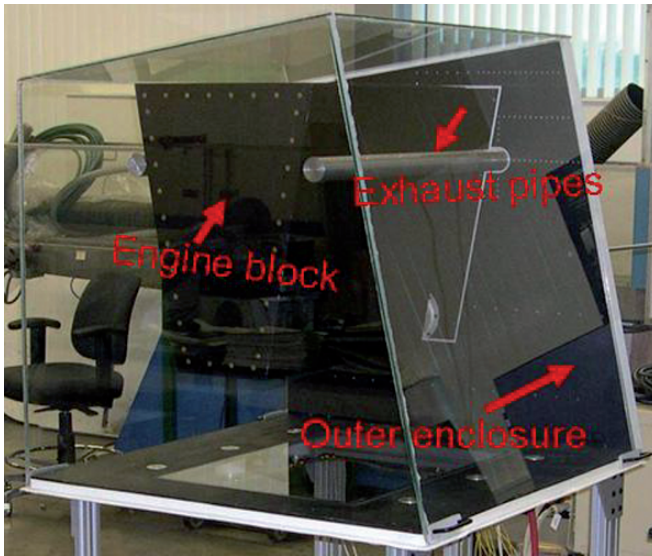


Fig. 1 Experimental setup

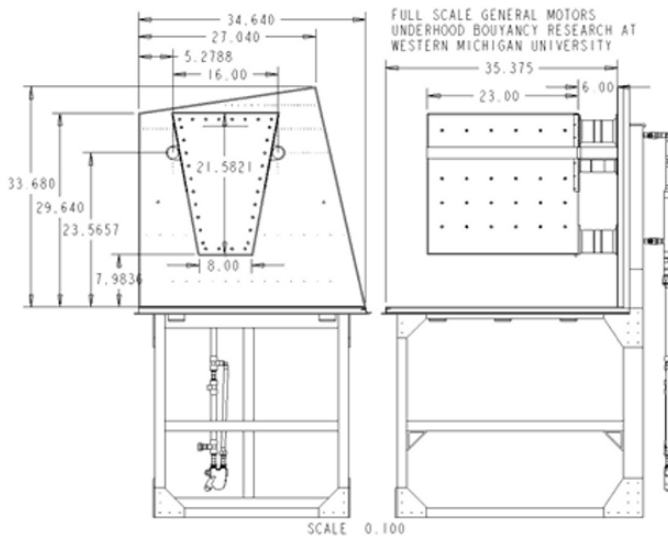


Fig. 2 Overall dimensions of the experimental underhood (units in inches)

2.1 Wall Boundary Temperature conditions

Two sets of boundary temperatures were recorded for the numerical validation purpose. The first is a fixed temperature boundary condition and the second is a time varying cool down temperature boundary condition. They are described as follows.

Fixed boundary temperature

There are three types of boundary surfaces in which temperature distributions need to be experimentally controlled. These three surface types are: exhaust cylinders, engine block and the outer enclosure. The surface temperature of each exhaust cylinders is kept at 600 °C by inserting a 3500 W cartridge heater into the core of the cylinder. The surface temperature of the engine block is kept at a nominal temperature of 100 °C by a specially designed water spray circulating system inside the engine block. A total of 116 thermocouples were buried under the skin of the engine block to monitor and record the actual temperature distribution on the engine block surfaces. As can be seen later, some surface areas, due to the proximity to the two exhaust cylinders, are inevitably hotter than 100 °C. This design is much simpler and more effective than that used in the quarter-scale setup [1]. No special device was used to keep the surface temperatures of the outer

enclosure constant. Instead, thermocouples were directly placed on the enclosure surfaces to record the final equilibrium temperature. The total number of thermocouple measuring points on the outer enclosure to map the temperature distribution is 451.

Time varying cool-down boundary temperature

When the vehicle is in thermal soaking condition, the surface temperature in each underhood component is changing with time. To model this cool-down thermal environment, a second set of temperature boundary conditions were recorded. Following the previously described fixed temperature boundary condition, the power used to maintain the temperatures for the exhaust cylinders and the engine block was shut off and the temperature readings were recorded for all the boundary surfaces for 45 minutes. The time history of the temperature cool-down data will provide time varying boundary conditions for another set of numerical validation and calibration. The experimental data of the transient cool-down boundary condition are documented in [4, 5] and are not presented here.

2.2 Air temperature and flow measurements

Air temperature measurement

The air temperatures are measured by thermocouples at 104 locations inside the engine compartment. These locations are on the mid-plane (at $y=0$) and their locations can be seen in Fig. 3.

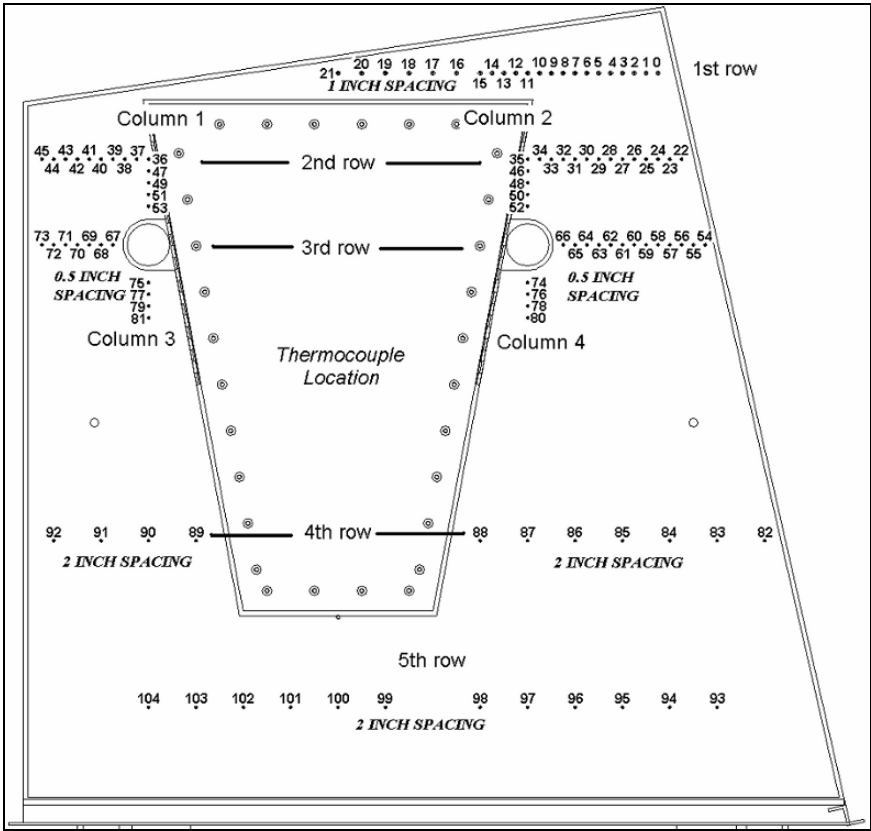


Fig. 3 Locations of the air temperature measurements

PIV flow measurement

The air velocity was measured by a conventional 2D PIV (Particle Image Velocimetry) system. The PIV setup can be seen in Fig. 4. There are three PIV test planes that correspond to mid-plane ($y=0$), front quarter plane ($y=-127$ mm) and back quarter plane ($y=+127$ mm). The camera field of view for these PIV tests was 12 cm by 12 cm in size. An interrogation area size of 32 x 32 pixels corresponding to physical size of approximately 4 mm x 4 mm provided adequate signal to noise ratio for repeatable results. Thus, to capture velocity field one region at a time, each vertical plane was broken up into 49 separate regions shown in Fig. 5. With this limited field of view, the measurement regions are overlapped to increase vector density and avoid discontinuities near the regions' borders. The regions are combined to show the flow statistics on an entire vertical plane.

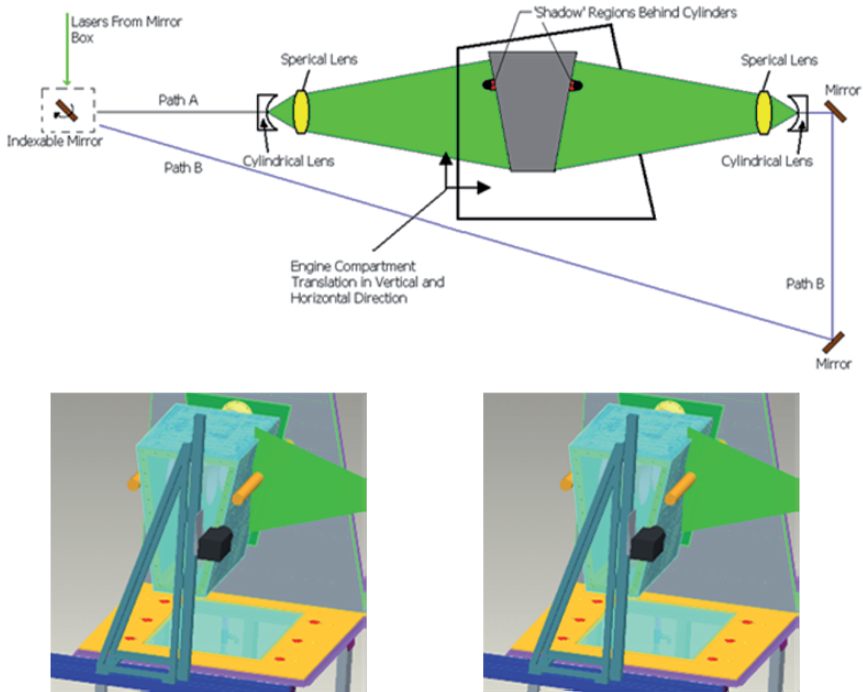


Fig. 4 PIV test setup

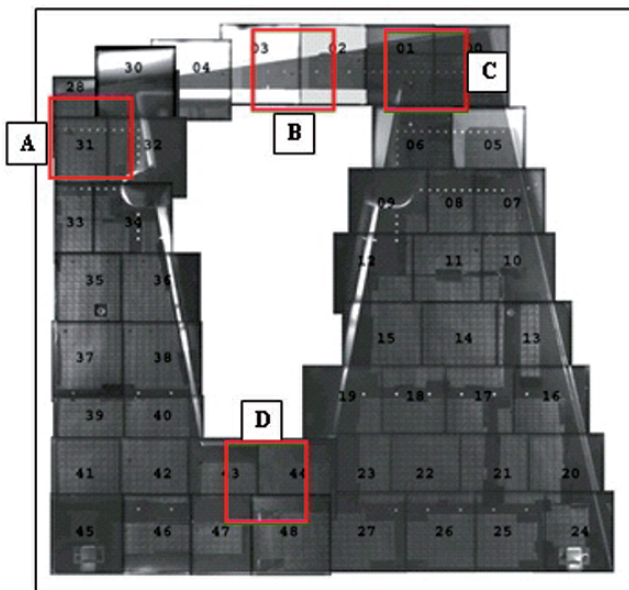


Fig. 5 Stitched overlapping region for PIV.

3 Experimental Results

The experimental results were categorized into three parts: (1). the surface temperature maps on the engine block and outer enclosure, (2). air velocity at the mid-plane, and (3). air temperature at the mid-plane. The surface temperature data are to be used as boundary conditions for numerical validation. The air temperature and velocity from the measurement at the mid-plane are the data to be used for numerical validation and comparison. Again the transient boundary temperature cool-down data described above are in large quantity and are not reported here. It is described in detail in [4, 5].

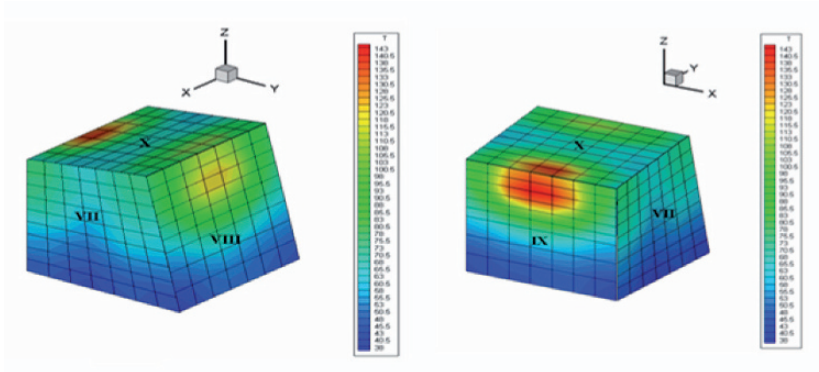


Fig. 6 Temperature maps for the outer enclosure

Table 1 Outer enclosure temperature statistics

	Glass Outer Surface Temperature °C			
	Front (VII)	Right (VIII)	Left (IX)	Top (X)
Mean	59.7	80.6	84.4	83.8
Std Dev	10.4	24.7	32.6	24.0
Max	77.7	124.4	147.2	146.4
Min	40.4	41.2	39.1	58.8
Range	37.3	83.2	108.0	87.6

3.1 Surface temperatures

The temperature maps on all the surfaces are recorded. These include exhaust cylinders, engine block, outer enclosures and the three pipes which provide the heating and cooling fluids and the thermocouple wiring in and out of the engine block. Figure 6 shows the surface temperature distribution of the outer enclosure. The temperature distribution is quite non-uniform with strong influence from the exhaust cylinders on both sides. Table 1 lists the temperature statistics of the outer enclosure. Figure 7 shows the temperature distribution for the surfaces of the engine block and Table 2 lists the temperature statistics. Again, the surface temperatures on the engine block are much higher than 100 °C due to the proximity to the hot exhaust cylinders. The average surface temperatures of the three pipes at the back of the engine block as seen in Fig. 2 are 95.7 °C for the upper pipe, 92.8 °C for the middle pipe and 74.8 °C for the lower pipe respectively.

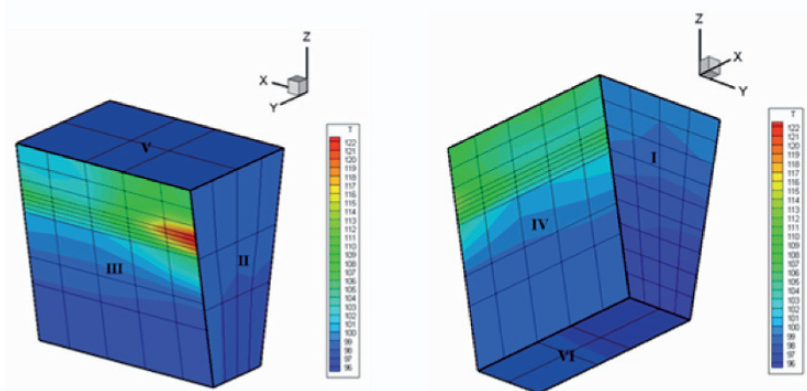


Fig. 7 Temperature maps for the engine block.

3.2 Air velocity at the mid plane

Figure 8 shows a time-averaged velocity contours at the mid-plane with close-up views from the upper left and right corners. Most of the flow activity occurs near the top of the compartment above the exhaust heaters and in the region between the top of the engine surface and the top enclosure wall. The maximum measured velocity magnitude is about 0.72 m/sec above the exhaust heaters. The flow field under the current flow configuration eventually evolves into an oscillating thermal plume above the exhaust cylinders. Figure 9 shows six thermal plume pictures in a cycle which has a frequency of 2.5 Hz. More discussions about the

transient behavior of the thermal plumes will be given when the numerical results are presented.

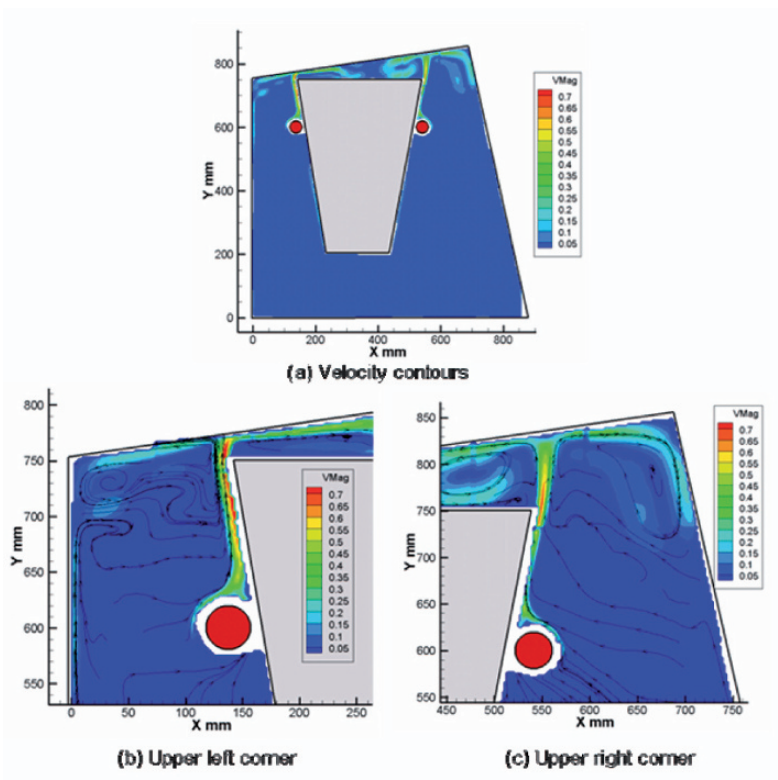


Fig. 8 Time-averaged velocity contours at mid-plane

Table 2 Engine block temperature statistics

	Front (I)	Back (II)	Right (III)	Left (IV)	Top (V)	Bottom (VI)
Mean	97.6	98.2	105.1	103.8	98.6	97.9
Std dev	1.4	0.5	7.0	4.2	0.1	0.5
Max	99.7	98.9	123.2	113.6	98.8	98.2
Min	94.3	97.2	97.6	98.2	98.4	97.1
Range	5.3	1.7	25.6	15.4	0.4	1.2

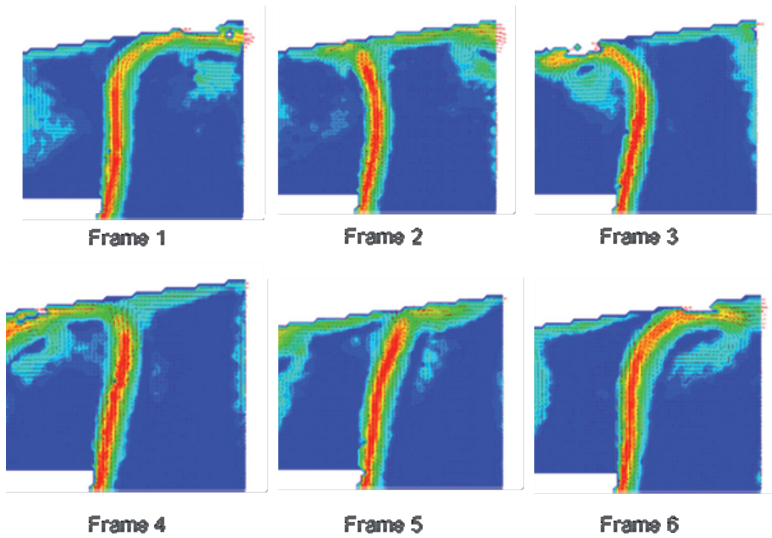


Fig. 9 Experimental instantaneous unsteady thermal plume on top of the right exhaust cylinder in a cycle

3.3 *Air temperature at the mid plane*

The location of the thermocouples at the mid-plane is shown in Fig. 3. There are 104 thermocouples used at the mid-plane with more distributed near and above the exhaust cylinders and the gap between the top engine block surface and the upper surface of the outer enclosure. The measured temperature distributions for rows 1 to 5 are shown in Fig. 10. These temperatures are to be compared with the numerical predictions and will be discussed later in the paper.

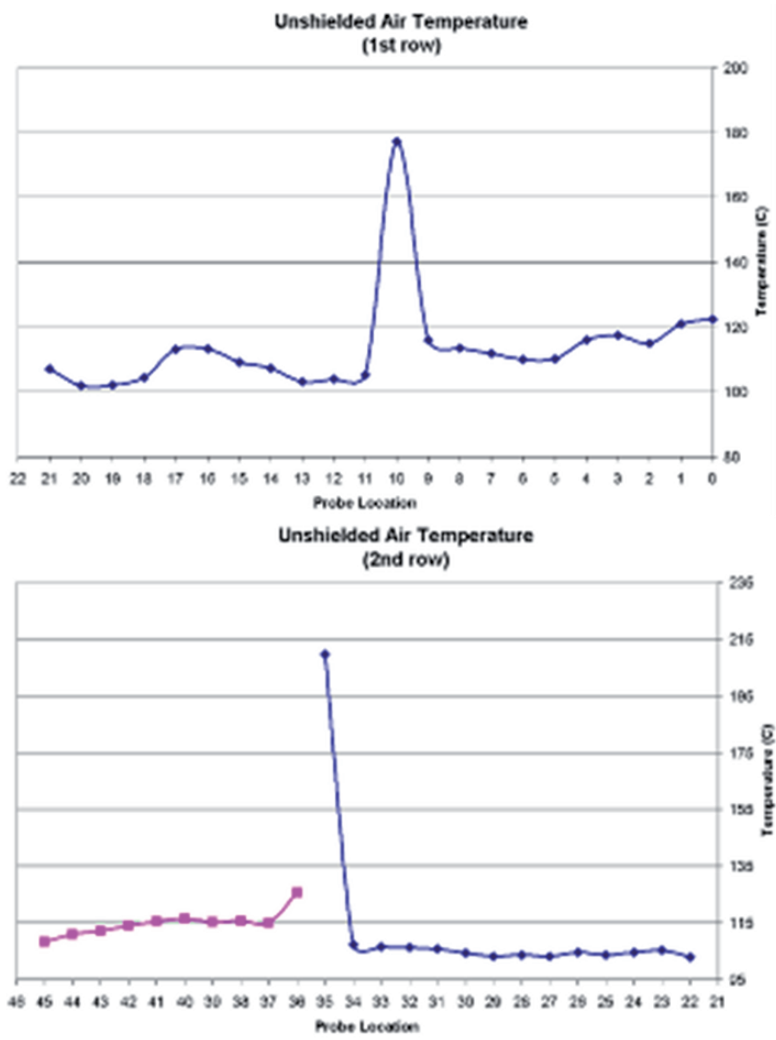


Fig. 10 Temperature profiles at various lines defined at the mid-plane.

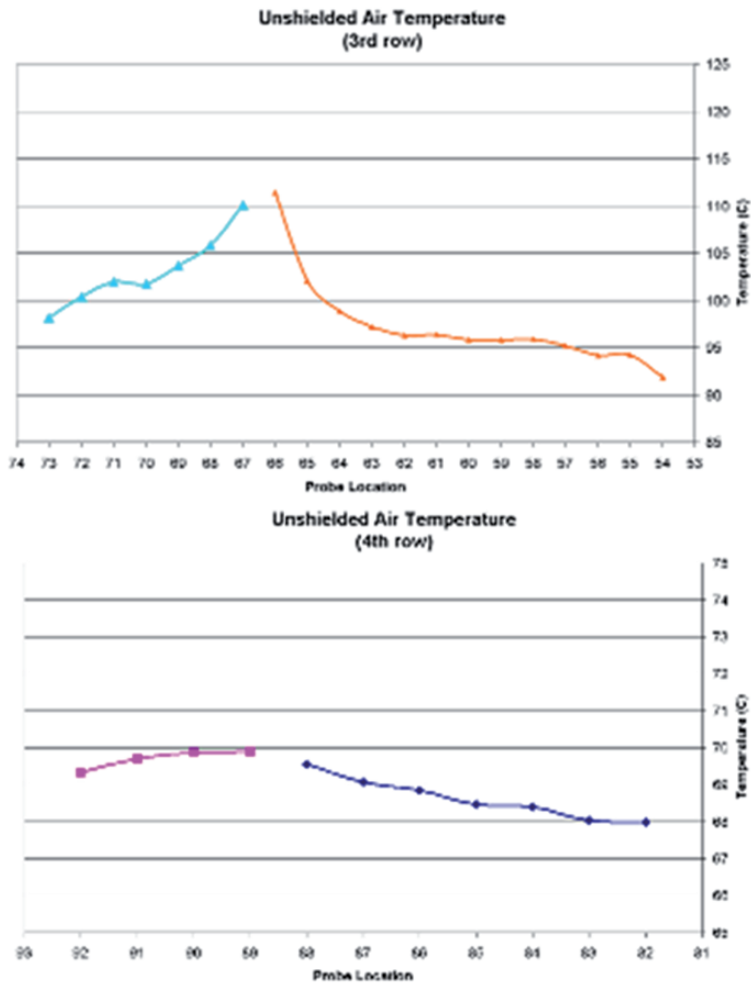


Fig. 10 (Continue)

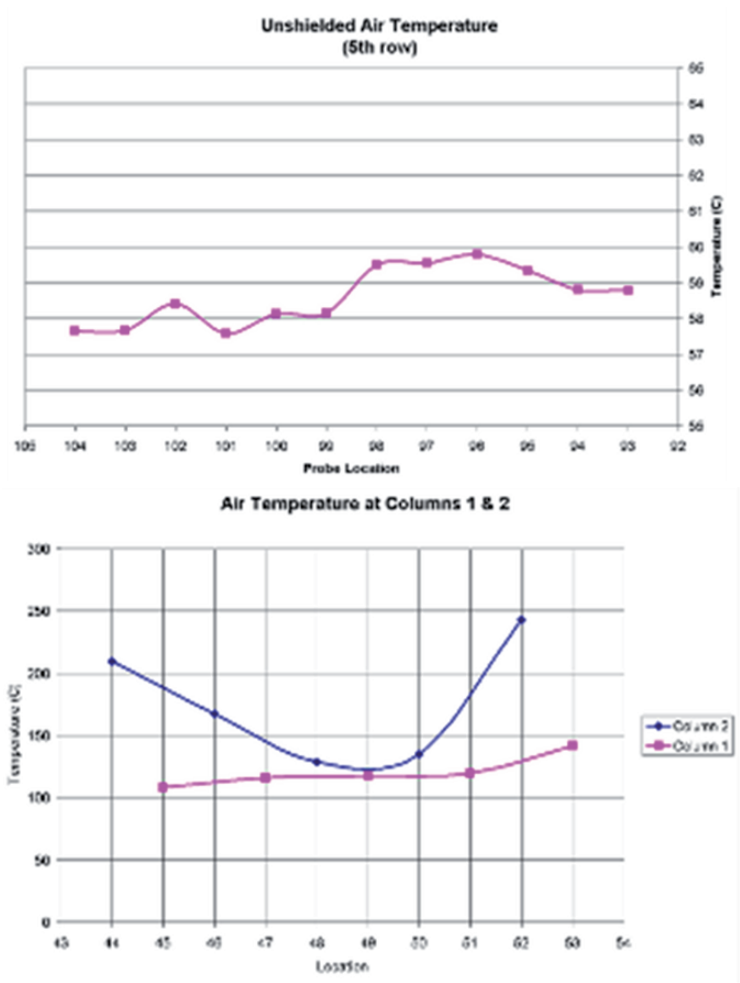


Fig. 10 (Continue)

4 Numerical Results

Numerical simulations were carried out with Fluent version 6.3.26 [3]. From the previous quarter-scale study [2], it was concluded that the 2nd order accuracy in both space and time is best suitable for the present simulation. It also demonstrates that a well prepared tetrahedral mesh can achieve a comparable accuracy with a prism layer mesh. The experiences gained from the detailed numerical study from the quarter-scale study [2] lead to the following mesh size arrange-

ment. The mesh size on the surface of the exhaust cylinders and engine block are 2 mm and 5 mm respectively. The total number of cells is 4.7 million tetrahedral cells. Much more cells were placed near the exhaust cylinders and the upper portion of the engine block to resolve the high temperature gradient and buoyancy induced flow field. The temperature boundary conditions on the surfaces of the engine block and the enclosure are directly obtained from the experimental data shown in Figs. 6 and 7, respectively (see also data in [4, 5]). The temperature on the surface of the exhaust cylinders was kept constant at 600 °C. A no-slip condition is used on all the wall surfaces. The RNG $\kappa - \varepsilon$ turbulence model with a non-equilibrium wall function is chosen to model the turbulence. The air dynamic viscosity and thermal conductivity vary with temperature by Sutherland law [3]. The range of air density variation in the domain was very significant due to the large temperature difference (~ 500 °C) and therefore the compressible ideal gas law was used to ensure the validity of the solution. A 2nd order implicit time marching scheme with sub-iterations was used to solve the flow and temperature fields from an initially quiescent flow field with a uniform temperature of 60 °C. The size of the time step reduced sequentially along the transient simulation process from 0.01 to 5×10^{-4} second to help obtain fast initial transient flow development and ensure capturing the accuracy of the fine scale flow motion toward the end of the simulation. The sensitivity of the time step to the accuracy of the transient motion will be discussed later in the paper.

The computational domain and the mesh in the mid-plane are shown in Fig. 11. The domain was set up exactly according to the experimental configuration which also includes the two bigger pipes used to support the engine block and to provide the plumbing supply to maintain the desired temperature of the engine block and the middle small pipe for thermocouple wires. The measured temperature maps on the wall boundaries were used to specify as the boundary conditions in the numerical simulation.

The simulation was carried out from a steady state run for 1000 iterations and then switched to the transient mode. The flow quantities were monitored for transient behavior at points 10 and 39 at the mid-plane as indicated in Fig. 12 (see also Fig. 3).

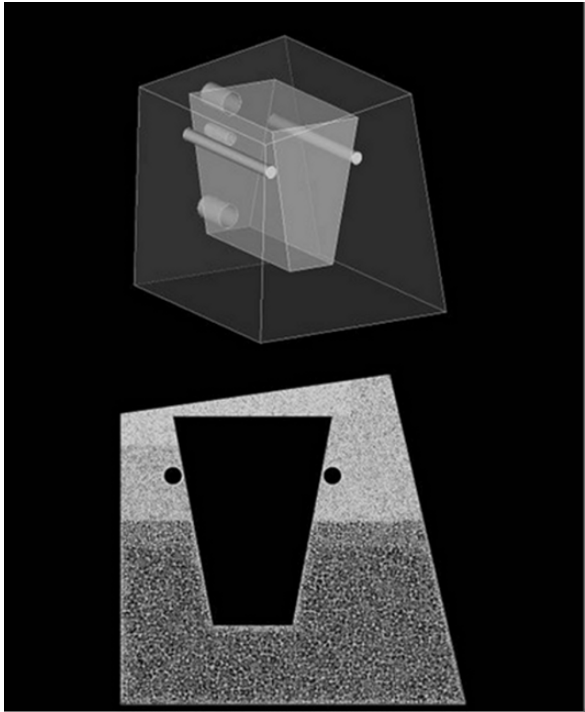


Fig. 11 The computational domain and the mesh at mid-plane.

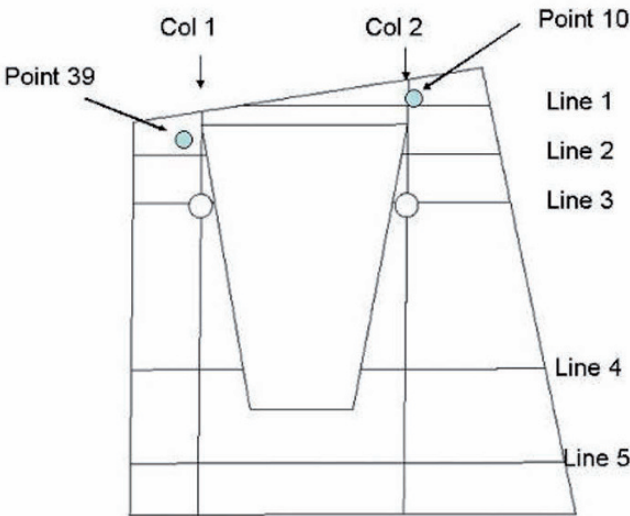


Fig. 12 Velocity and temperature profiles recorded for comparison.

4.1 Transient behavior of the flow

Figure 13 shows the time histories of the velocity magnitude at points 10 and 39. Due to the slope of the upper outer enclosure, the upper right corner has more room for the flow to develop. This can be seen in Fig. 13. It shows that the velocity magnitude at point 10 is much higher values than that at point 39. The time history also shows that the flow fields in the two monitor points are relatively quite for the first 10 seconds. The region at and around the point 39 starts to respond to the effect of the thermal plume originated from the left exhaust cylinder between $t = 5$ and 15 seconds. The velocity at point 39 stays flat at lower than 0.1 m/s for the next 30 seconds before it turns violent and exhibits a very regular periodic motion with an amplitude of around 0.1 m/s. The time history of the velocity magnitude at point 10 behaves similar to point 39 except that the velocity magnitude at point 10 is much larger than that at point 39.

To gain more insights of the final oscillating pattern, the time histories of the velocity and temperature at these two points for the last 5 seconds are shown in Fig. 14(a) and Fig. 14(b). They are clearly developing into a periodic motion as suggested by the experiment [4, 5] shown above. Figure 15 shows the power spectral density of the velocity at point 10. It clearly reveals a dominant frequency at about 2.53 Hz (the time period $T=0.395$ s). The frequency recorded in the experiment shown in Fig. 9 is 2.5 Hz. The agreement with the current numerical prediction is very satisfactory. This comparison is also listed in Table 3. The ability to capture the transient accuracy during system integration is extremely important in predicting the correct peak temperature during the soaking stage to prevent part failure due to inadequate heat protection.

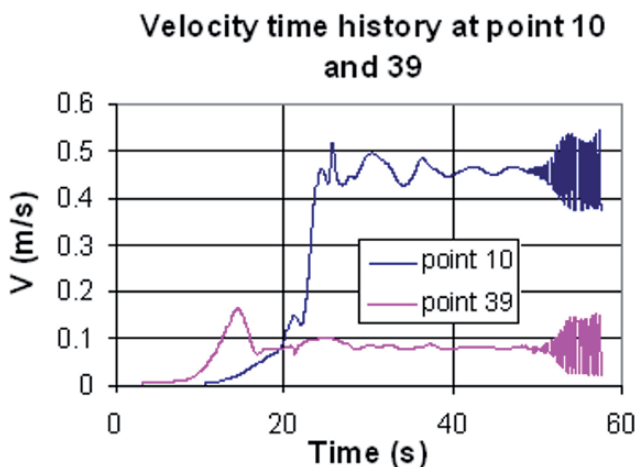


Fig. 13 Time histories of the velocity magnitude at points 10 and 39

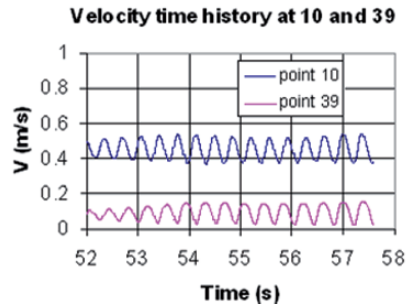


Fig. 14a Close-up view of the oscillating pattern for velocity at points 10 and 39

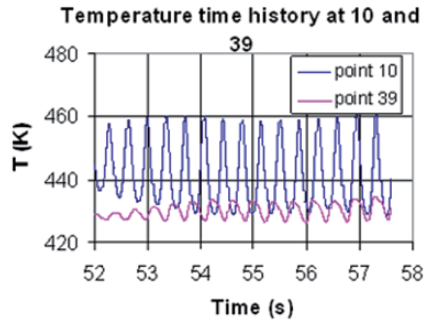


Fig. 14b Close-up view of the oscillating pattern for temperature at points 10 and 39

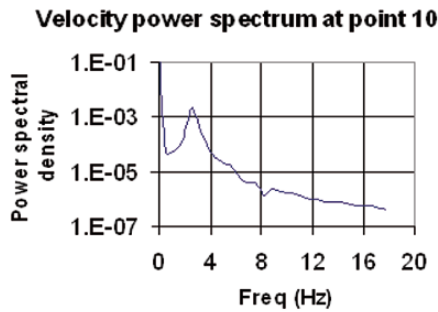


Fig. 15 Velocity power spectrum at point 10.

Table 3 Comparison of the oscillating frequency of the hot thermal plume

	Frequency (Hz)	Period (second)
exp	2.5	0.4
Fluent	2.53	0.395

4.2 Sensitivity of the time step size

As mentioned earlier, the size of the time step reduced sequentially along the transient simulation process from 0.01 to 5×10^{-4} second to help obtain fast initial transient flow development and ensure capturing the accuracy of the fine scale flow motion toward the end of the simulation. A separate study was conducted to investigate sensitivity of the time step size to the solution accuracy. Figure 16 shows the results of this study. It was found that for the current flow, if a constant large time step of 0.01 second was used, the thermal plume never developed a correct periodic motion as indicated by the velocity time history at point 10. When compared to Fig. 13 which shows a correct velocity time history using time step of 5×10^{-4} second, the velocity curve at point 10 begins to show development of periodic motion as early as around 50 seconds, the velocity time history at the same point in Fig. 16 never develops a periodic motion up to 67 seconds when the time step of 0.01 second is used. It was found that the reduction in time step at least by a factor of 10 (time step of 0.001 second) would allow the flow to establish a converged flow behavior. The velocity time history for a time step of 0.001 second is shown in Fig. 16 by restarting the simulation from the solution of 0.01 second obtained at time=65 seconds. It is very clear that a converged (in a transient sense) periodic motion begins to develop right after using the time step of 0.001 second.

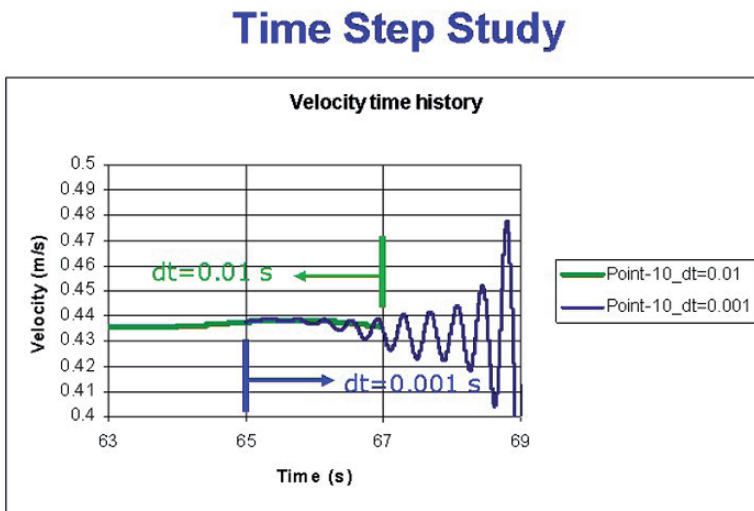


Fig. 16 Time history plot for the velocity at point 10 with two time step sizes during the final stage of simulation.

4.3 Buoyancy flow characteristics

The Grashof number, Gr_L , is usually used to characterize the buoyancy flow. The definition of the Grashof number is expressed as follows:

$$Gr_L = \frac{g\beta (T_s - T_\infty) L^3}{\nu^2}$$

where g is the gravity, β is the thermal expansion coefficient, ν is the kinematic viscosity, L is the length scale, T_s is the surface temperature, and T_∞ is the temperature in the flow-field. Due to the complexity of the geometry, two Grashof numbers are identified. As shown in Table 4, the two Grashof numbers are based on two length scales, one with the slanted engine block wall and the other with the exhaust cylinder. T_∞ is based on the average flow temperature close to the heat sources as indicated in Table 4. The estimated range of the Grashof numbers indicates that it is very close to the turbulent regime according to the classic cylinder flow analysis [6].

Table 4 Buoyancy flow characteristics

	Right engine block	Right exhaust
$g \text{ (m/s}^2\text{)}$	9.81	9.81
$\beta \text{ (1/K)}$	0.0026	0.0026
$T_s \text{ (}^\circ\text{C)}$	123	600
$T_\infty \text{ (}^\circ\text{C)}$	104.3	104.3
$L \text{ (m)}$	0.557	0.0457
$\nu \text{ (m}^2\text{/s)} @ T_f=0.5(T_s + T_\infty)$	2.77×10^{-5}	5.4925×10^{-5}
Gr=	1.0569×10^8	4.0×10^5

4.4 Time-averaged temperature and velocity

Figure 17 shows snapshots of both temperature and velocity contours at mid-plane. The velocity contours shown in Fig. 17 qualitatively resemble the experimental data shown in Fig. 8. As mentioned earlier, the flow exhibits a periodic pattern as both suggested by the test and numerical simulation. Therefore to compare with the experimental data for both the temperature and velocity components, the flow results from the numerical simulation over many cycles into the oscillating

periods are time-averaged. The time-averaged temperature and velocity components are compared with the measurements in five rows and two columns defined in Fig. 12. Figure 18 shows the temperature profiles from the current simulation for 5 horizontal rows. The symbols represent the measured data reported in [4, 5].

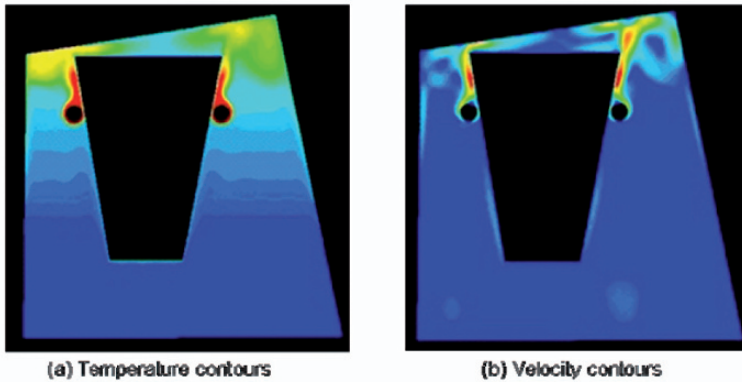


Fig. 17 Snap shots of the temperature and velocity contours at the mid-plane

The temperature profiles are shown in Fig. 18. The predicted temperature profile in line L1 which is in the gap between the upper surface of the engine block and the outer enclosure follows the trend quite well with the test data except that it under-predicts the peak value near the right enclosure surface. Similarly the profile in line L2 on the right part shows the correct trend with the test data but misses the peak by about the same amount. The temperature discrepancy around this small region close to the engine block was thought to come from the under-resolved boundary temperature mapping from the experimental data where only finite number of thermocouples was installed. More thermocouples in the future measurement should provide better boundary conditions for numerical validations. Figure 19 shows the temperature profiles along two vertical columns. The figure shows very clearly that the temperature gradient near the exhaust cylinders is very high.

The velocity components are compared with the PIV measurements along the same five horizontal rows and two vertical columns. Figure 20 shows the W (vertical) velocity component on those five rows. The trend in the velocity profiles is well captured. It should be noted that the velocity magnitude is very small and is in the order of 0.7 m/s. Figure 21 shows the U (horizontal) velocity component along the two vertical columns. The trend of the profiles is also very well predicted. However, it can be seen in Figs. 20 and 21 that the numerical results under-predict the peak values for both U and W components. This under-prediction is very consistent to the temperature profiles discussed in the previous paragraph. A higher degree of boundary temperature mapping from the measurement with more thermocouples should be able to improve the prediction accuracy.

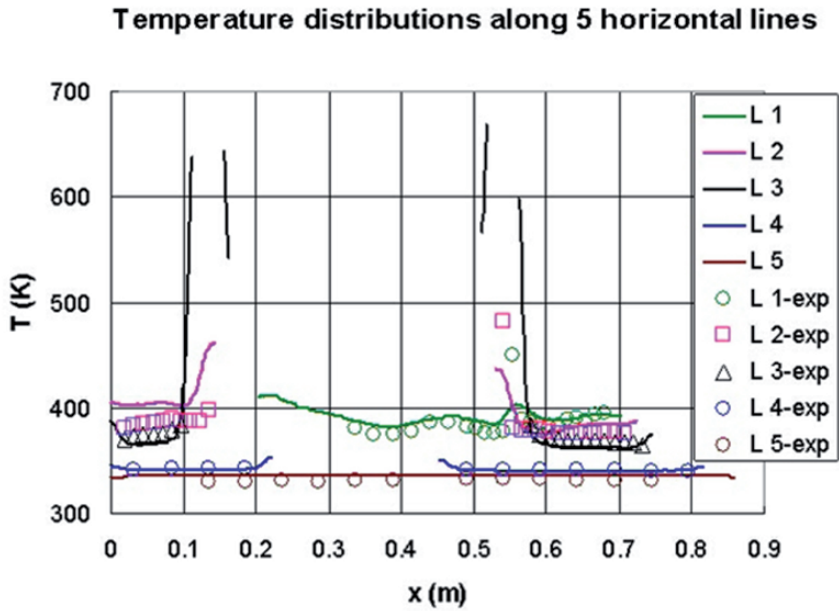


Fig. 18 Temperature profiles comparison with the measured data in five rows.

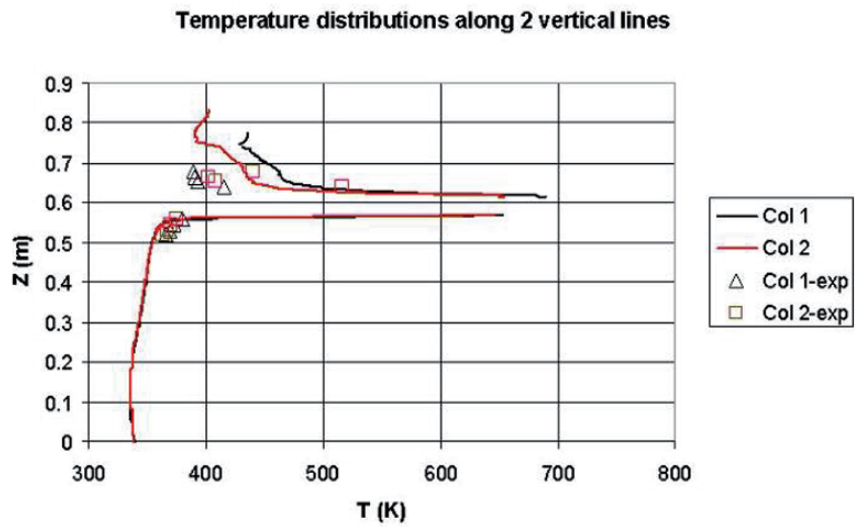


Fig. 19 Temperature profiles comparison with the measured data in two columns.

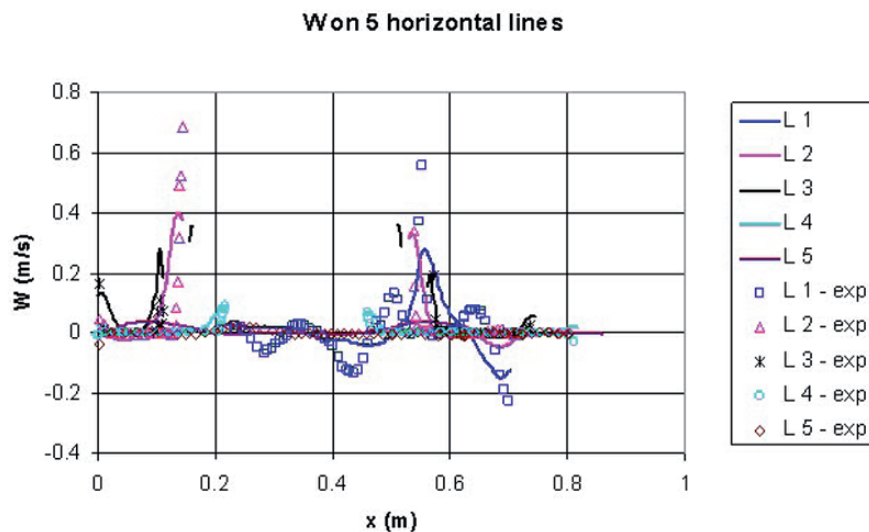


Fig. 20 W velocity component profile comparison with the measured data in five rows

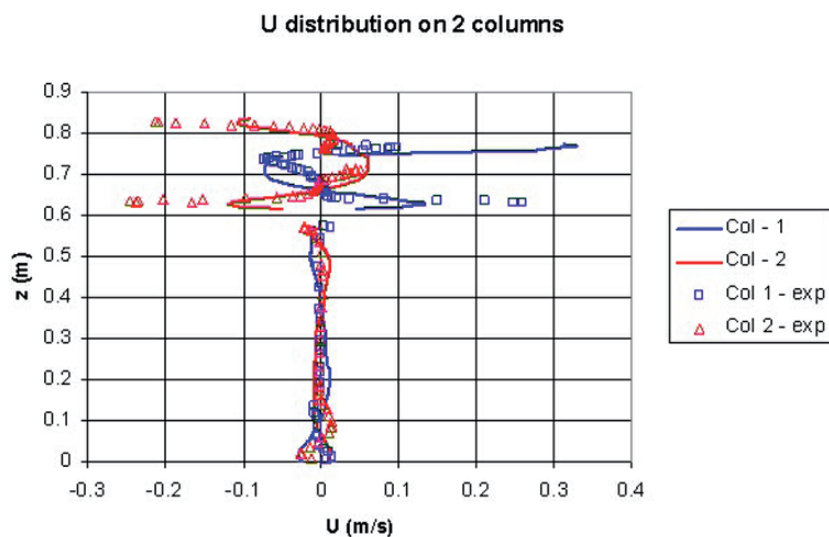


Fig. 21 U velocity component profile comparison with the measured data in two columns

5 Conclusions

Both experimental and numerical approaches are used to study the buoyancy driven flow in a simplified underhood. The simplified underhood geometry consists of an enclosure, an engine block and two exhaust cylinders mounted along the sides of the engine block. The experiment measures both temperature at the mid-plane and PIV velocity field at the mid-plane and 2 off-center planes. The wall temperature distributions along all the boundary surfaces are also recorded to be used for numerical validation. The numerical simulation predicts the transient thermal plume right above the exhaust cylinders very well with the predicted oscillating frequency at 2.53 Hz which is very close to the test data of 2.5 Hz. The contours of the flow and temperature fields show well-captured thermal plumes right above the exhaust cylinders, recirculation vortices around two upper corners and the gap between the engine block and the upper enclosure surface. The time-averaged air temperature and velocity profiles at the mid-plane for five horizontal rows and two vertical columns are compared to the measured data. The temperature values, in general, are in close agreement with the test data except the numerical results under-predict the peak temperature in the gap region between the top surface of the engine block and the outer enclosure. The under-predicted temperature and velocity near the engine block are believed to come from the under-resolved boundary temperature distributions. A more detailed boundary temperature mapping from the measurement can improve the current accuracy. The predicted maximum air velocity scale is in the order of 0.7 m/s which is extremely small and very hard to obtain accurately in the measurement. Nonetheless, the numerical predictions capture the flow motion quite well when compared to the measured velocity profiles.

A transient cool-down air flow and temperature measurement was also conducted in the experiment. The time history of the 45-minute temperature boundary condition was recorded to be used later as transient boundary conditions for future numerical simulations. A limited number of air temperature and velocity data above the thermal plume are recorded for 45 minutes for later comparison. The numerical study of the transient cool-down test condition will be presented in a separate paper.

Acknowledgments

The authors would like to express gratitude for the funding support by Jim Welton at GM Global Engineering. Additional thanks to Rose McCallen, Fred Browand, Jim Ross and ECI for the organization of The Aerodynamics of Heavy Vehicles II conference.

References

- [1] Merati P, Cooper N, Leong C, Chen K-H and Johnson J, “Investigation of the Buoyancy Driven Flow in A Simplified Underhood - Part I, PIV and temperature Measurements”, SAE Paper 2006-01-1608, *SAE Congress 2006*, Detroit, MI.
- [1] Merati P, Cooper N, Leong C, Chen, K-H and Johnson J, “Investigation of the Buoyancy Driven Flow in A Simplified Underhood – Part I, PIV and Temperature Measurements”, SAE Paper 2006-01-1608, SAE Congress 2006, Detroit, MI.
- [2] Chen K-H, Johnson J, Merati P, Cooper N and Leong C, “Investigation of the Buoyancy Driven Flow in A Simplified Underhood - Part II, Numerical Study”, SAE Paper 2006-01-1607, *SAE Congress 2006*, Detroit, MI.
- [3] Fluent 6.3.26, Fluent Inc. 2007
- [4] Merati P, Leong C and Sakuma Y, “Investigation of the Buoyancy Driven Flow in a Simplified Full-Scale Underhood – PIV and Temperature Measurements”, Mechanical and Aerospace Engineering, Western Michigan University, MI. July 13, 2006.
- [5] Merati P, Leong C, Chen K-H and Johnson J, “Investigation of the Buoyancy Driven Flow in a Simplified Full-Scale Underhood – PIV and Temperature Measurements”, to be published in Springer-Verlag Lecture Notes in Applied and Computational Mechanics :*The Aerodynamics of Heavy Vehicles II: Trucks, Buses and Trains*, July, 2008.
- [6] Holman J *Heat Transfer* fourth edition, 1976, McGraw-Hill, Inc.