

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

Ramesh Pankajakshan, Brent Mitchell & David L. Whitfield

University of Tennessee SimCenter at Chattanooga, USA
ramesh-pankajakshan@utc.edu

Abstract Computational Fluid Dynamics (CFD) simulations were performed to evaluate drag reduction devices on a modified full-scale version of the Generic Conventional Model (GCM) geometry. All simulations were performed with a moving ground plane and rotating rear wheels. A trailer base flap simulation was performed for comparison with drag reduction data from wind tunnels and track and road tests. A front spoiler and three mud-flaps with modest drag reduction potential were evaluated in view of their higher probability of adoption by truck fleet operators.

Introduction

Aerodynamic drag accounts for approximately half the fuel consumption of a class 8 truck traveling at freeway speeds and even a 1% reduction in the drag value can yield cost savings significant enough to initiate adoption by large trucking companies. While a large number of effective drag reduction devices have been developed and are commercially available [1], uptake in the trucking industry has been relatively insignificant due to a veritable thicket of infrastructure, legislative, maintenance, staffing and aftermarket issues which have rendered them cost ineffective to a large section of trucking companies. A good example of such technology is the base flap which has seen slow adoption in spite of drag reductions in excess of 10%. While there clearly is the potential for substantial drag reduction in the base region of the trailer, any changes to this area of the truck have to be made in light of severe restrictions imposed by existing infrastructure and operational norms. The underbody of the truck offers an area with less potential but with the kind of restrictions which are more amenable to engineering solutions. Moreover, drag reduction devices which target consumables like mud flaps are more likely to be tried out and adopted since they often need to be changed due to wear and tear. The underbody is however by no means a *tabula rasa* on

which the aerodynamicist may impose his/her will. There are restrictions in terms of ground clearance and ease of access for maintenance and inspection which have been the bane of extant devices such as side skirts. There are also less obvious issues such as the effect of any drag devices on brake cooling flows as well as splash and spray.

Simulations

Flow Solver

Simulations were performed using the UT Tenasi flow solver [11] which is an Unsteady Reynolds Averaged Navier Stokes (unRANS)/Discrete Eddy Simulation (DES) code operating on unstructured grids with a finite volume node centered flux formulation. The incompressible solver used for slow speed flows uses the artificial compressibility approach of Chorin[2]. The 1-equation Menter SAS model in DES mode [8] was used for all the simulations. Integration was performed to the wall using grids generated to produce off-wall y^+ values less than unity. The solver and grid generation approaches used in this work were validated using the Ground Transportation Systems (GTS) [3] and Generic Conventional Model (GCM) [13] experimental datasets. Pressure predictions were found to correlate well with experimental measurements over most of the geometry with the exception of the trailer base region [12].

Geometry and Grid Generation

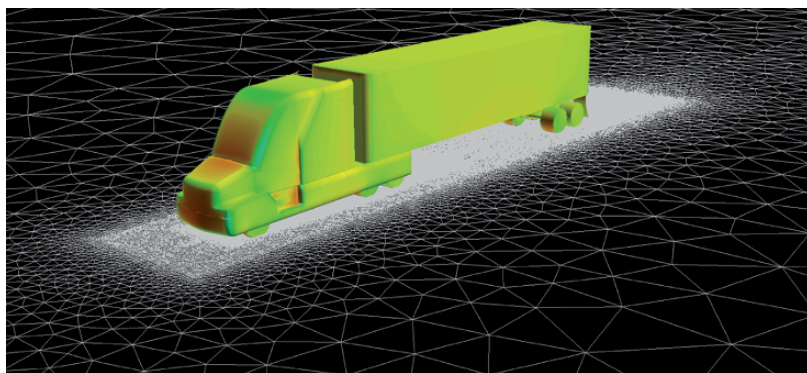


Fig. 1 GCM geometry and ground plane grid

A modified version of the GCM geometry (Fig. 1.) scaled for full-scale Reynolds numbers was chosen as the base geometry for the simulations in this study. The modifications include the addition of cab extenders and lowering of the geometry to allow the wheels to make contact with the ground plane. The support struts used for mounting the geometry in the wind-tunnel geometry were also removed. Fillets had to be introduced at the contact line between the wheel and the ground plane in order to facilitate the grid generation process.

The commercial grid generation package GridGen was used for generating a tetrahedral mesh for the GCM geometry. An in-house code called HUGG [6] was used to insert viscous layers. The off-wall spacings and growth parameters used were based on y^+ constraints and experience gained in the course of the validation process. The grids used here have approximately 1.1 million surface elements on the GCM surface and 18 million control volumes in the field. Total time to solution for each case was 3 days on 200 processor cores (Intel Xeon 5160, 3.0 Ghz EM64T, 4MB cache, dual core).

Flow Conditions

The flow simulations were conducted at full-scale i.e. for a Reynolds number of 5.15 million based on a trailer width of 8.5 feet and a road speed of 65 mph. All simulations used a moving ground plane matching the road speed and rotating rear wheels on the trailer. Rotating wheels were incorporated into the simulation through a slip boundary condition which has been shown to work well for smooth wheels [10].

Wind-averaged drag simulations were conducted assuming a 7 mph crosswind at six (15, 45, 75, 105, 135 and 165 degrees) orientations with respect to the direction of travel of the truck. The wind-averaging procedure follows the method of Ingram [5] as described in [9] except that the drag forces are compared in the form of non-dimensional force coefficients calculated using the road speed as reference.

Results

Effect of Spinning Wheels

In order to evaluate the effect of spinning wheels on the drag of the truck, the 45 degree crosswind case with mud-flaps was rerun with the spinning wheel

boundary condition turned off. It was found that with stationary wheels, the drag increased by 5.4% over the spinning wheel case, with more than half the increment being due to an increase in the drag on the axle-wheel assembly as shown in Figure 2. Increased drag on the mud-flaps and trailer contributed to 27% and 15% of the total respectively.

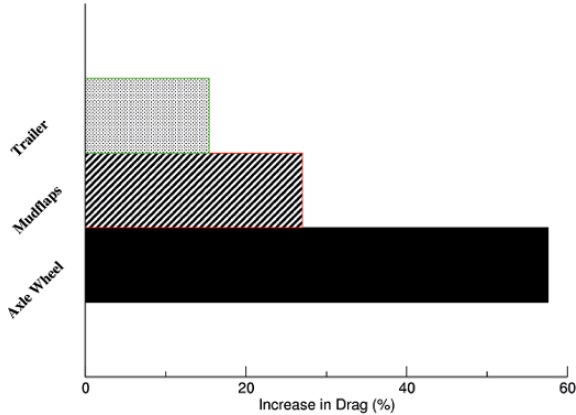


Fig. 2 Contributors to the drag increase caused by non-spinning wheels

Cab Extenders and Splitter Plates

The base GCM geometry which does not have cab extenders exhibits large drag increases in the presence of crosswinds. This effect can be almost totally mitigated by the addition of cab extenders or splitter plates on the front face of the trailer. Neither of these devices has any effect in the absence of crosswinds. Adding both devices provides no additional drag benefit. Since there are more trailers than tractors, cab extenders are probably the more economical option and they seem to have been universally adopted. The GCM model with cab extenders was chosen as the base model against which all drag reduction devices would be compared.

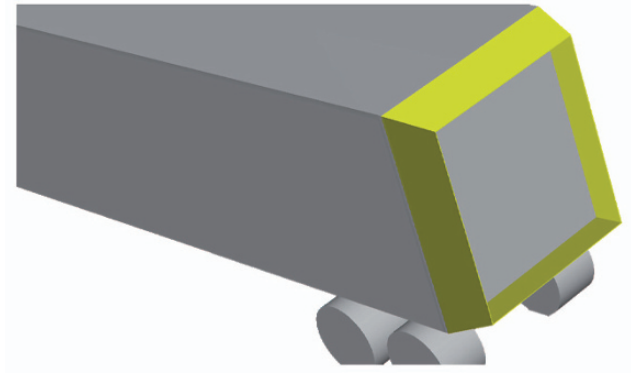


Fig. 3 Geometry of base flaps

Base Flaps

The base flaps consist of 4 panels inclined at an angle of 15 degrees to the axis of the truck as shown in Fig. 3. Each panel is one-quarter of the width of the trailer or approximately 2.1 feet. This device has been extensively studied by other researchers and a commercial product is available as well. This case was run to ensure that the drag reduction from this study was in qualitative agreement with the results in the literature. The computed wind-averaged drag reduction of 15% compares favorably with tunnel scale model tests (~14%) [13], track tests (~8.4%)[1] and road tests (~12% & ~20%)[7] with full trucks. The wind tunnel data was for the GCM geometry and is therefore in close agreement. The track and road tests were for actual class 8 trucks and the drag numbers are derived from fuel savings assuming that the percentage fuel saving was half the drag reduction percentage.

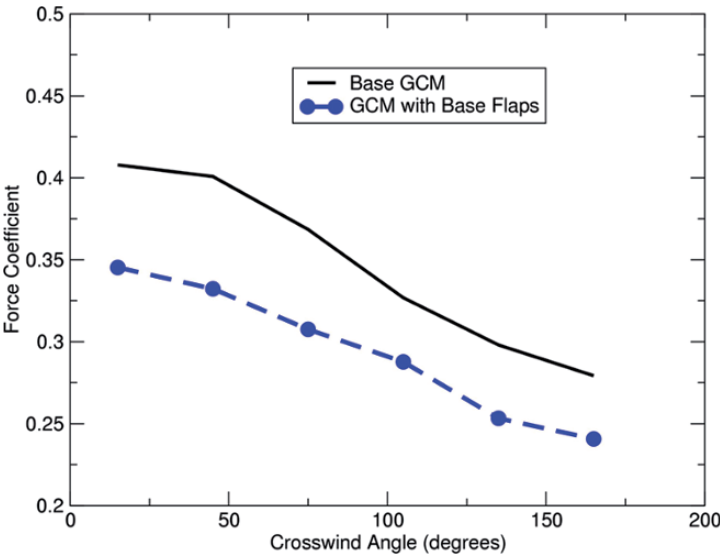


Fig. 4 Variation of axial force coefficient with crosswind angle

The drag force coefficients at the various crosswind angles are shown in Fig. 3. The effectiveness of the base flap was not particularly sensitive to the direction of the crosswind though it performs slightly better in head winds.

Front Spoiler

Front spoilers have been used in cars for a few decades to reduce underbody drag and improve cooling airflow but their effectiveness depends on careful sizing of the spoiler height [4]. The spoilers used in this study were approximately 3 inches in height and were located under the cab as shown in Fig.5.

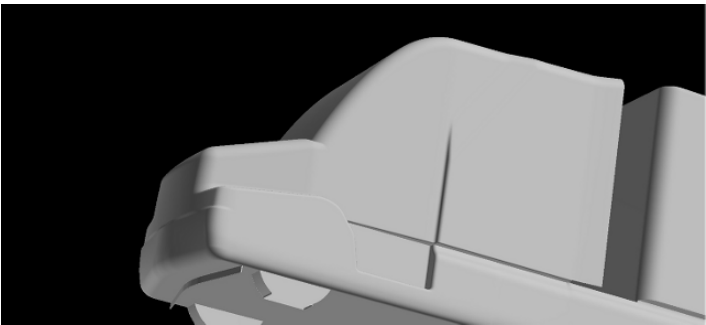


Fig. 5 Geometry and location of front spoiler

The front spoiler increased the wind-averaged drag on the GCM model by 0.2%. This implies that in an actual truck with front axles and other drag inducing components in the underbody region, the front spoiler could lead to small drag reductions. This is based on the fact that the drag increase due to the spoiler was 3.8% while it reduced drag on the rest of the truck by 3.6% even for a smooth underbody without front axles. Lowered velocities in the underbody region account for the reduced drag on the tractor and wheel assembly as well as a portion of the reduced drag on the trailer. The majority of the trailer drag reduction comes from lowering of the pressure on the front face of the trailer due to changes in flow over and around the tractor. The drag values at the various crosswind angles shown in Fig. 6b are also encouraging since the spoiler actually reduces drag when the truck is experiencing a head or tailwind. It might also be possible to modify the shape and/or height of the spoiler to reduce or eliminate the drag increase for non-parallel crosswinds. An accurate evaluation of this device will need simulations using a substantially more realistic truck model with underbody components and accurate capturing of the under-hood flow.

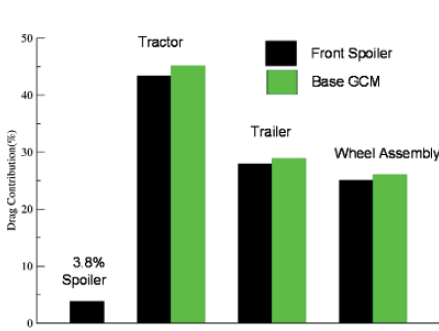


Fig. 6a Drag contributions of truck components

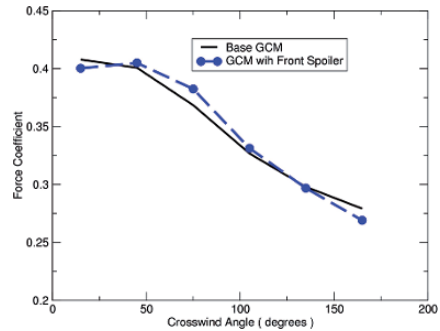


Fig.6b Variation of axial force coefficient with crosswind angle

Mud Flaps

Simulations were conducted to study the drag contribution of rear trailer mud flaps to the total drag and to look at ways in which this could be reduced. Full mud flaps which extend to the bottom of the trailer and half flaps that extend to the top the wheels as shown in Fig. 7a and Fig. 7b respectively were studied. Full flaps are usually used on trucks that transport shipping containers as well on “pup” trailers. Half flaps are more common and are usually installed on most dry freight trailers.

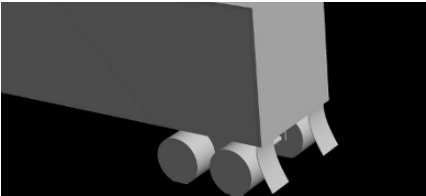


Fig.7a Geometry of full mud-flaps

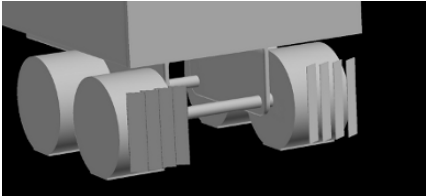


Fig.7c Geometry of half slats

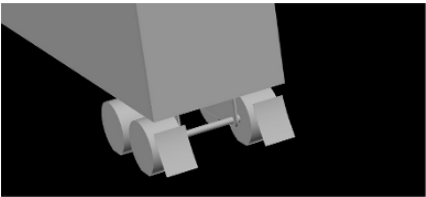


Fig.7b Geometry of half mud-flaps

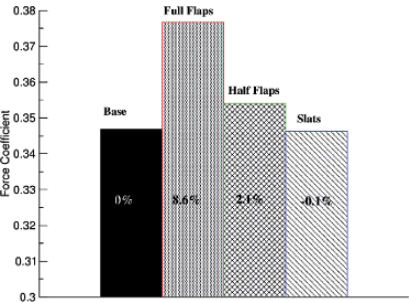


Fig.7d Drag force coefficients of trucks with various mud-flaps

The geometries of both flaps assume the presence of an anti-sail bracket with some curvature of the flaps beyond the device. A half slat (Fig. 7c) was studied as a possible low drag alternative for half flaps. The slats are inclined at an angle of 45 degrees to the flap plane and overlap each other to disallow straight flow through them. The wind-averaged drag value comparisons for the 3 components in Fig. 7d show that full flaps extract a heavy penalty by adding 8.6% to the drag while the half-flaps add only a modest 2.1%. The additional drag caused by half flaps can be completely eliminated by the use of half slats. Fig. 8a shows that the mud-flap and trailer account for 78% and 74% of the total drag increase while the wheel assembly experiences lower drag which accounts for 52% of the overall increase. The drag on the half-slats are almost identical to that on the base GCM for most crosswind angles as can be seen in Fig. 8b.

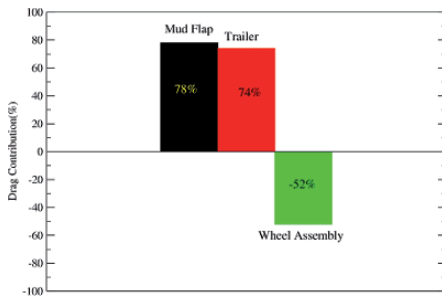


Fig. 8a Drag increase contributions for truck with full rear flaps

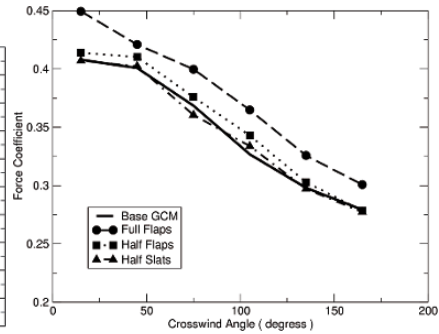


Fig. 8b Variation of drag with crosswind angle for truck with rear mud-flaps

Conclusions

The simulations with and without spinning wheels indicate that a more realistic treatment of the wheels in trucks is needed particularly when evaluating components in close proximity to the wheels. The question of whether a simple slip boundary condition based smooth wheel is sufficient to capture the change in flow characteristics caused by a real wheel with treaded tires and wheel holes cannot be answered without experimental studies and validation.

The base flap simulations demonstrate that the flow solver and grid resolution used in this study can produce a reasonable estimate of the relative magnitude of the drag reduction in the trailer base region in spite of the fact that validation studies show that CFD methods have a clear weakness in predicting flow features in this region. The implication is that a gross feature like the averaged pressure over the trailer back is being captured reasonably well while the smaller scale features are being missed. There is no guarantee that this is likely to be true for any and all devices that work in the base area and therefore simulations should be looked upon with skepticism in the absence of supporting experimental evidence.

The front spoiler simulation did not demonstrate any drag reduction for the GCM truck model. However the fact that the spoiler added only a negligible amount of drag indicates that the device is likely to reduce drag by a few percentage points for real truck with a significant amount of blockage in the underbody area.

A complete analysis of the mud-flaps would require the solution of a coupled fluid-structures problem downstream of a fully detailed rotating wheel. The current study has simplified the problem considerably in order to make it tractable and the results indicate that mud-flaps, particularly full flaps, are extracting a rather high toll in terms of their drag contribution. The half slat simulations indicate that simple changes to the mud flaps might be sufficient to mitigate their drag contributions. Other components in that area such as the rear impact guard, the

brake light panels and the license plate could be redesigned to perhaps yield small drag savings as well.

References

1. Browand F, Radovich C, & Boivin M (2005) Fuel Saving by Means of Flaps Attached to the Base of a Trailer: Field Test Results. SAE Paper 2005-01-1016.
2. Chorin AJ (1967) A Numerical Method for Solving Incompressible Viscous Flow Problems. *J Comput Phys* 2:12-26
3. Croll RH, Gutierrez WT, Hassan B, Suazo JE & Riggans AJ (1996) Experimental Investigations of the Ground Transportation Systems (GTS) Project for Heavy Vehicle Drag Reduction. SAE Paper 96-0907.
4. Hucho W-H (1998) *Aerodynamics of Road Vehicles: From Fluid Mechanics to Vehicle Engineering*, SAE International.
5. Ingram KC (1978) The Wind-Averaged Drag Coefficient Applied to Heavy Goods Vehicles. Transport and Road Research Laboratory Supplementary Report 392.
6. Karman S (2006) Unstructured Viscous Layer Insertion Using Linear-Elastic Smoothing. AIAA Paper 2006-0531.
7. McCallen R et al (2006) DOE's Effort to Reduce Truck Aerodynamic Drag through Joint Experiments and Computations.
www.eere.energy.gov/vehiclesandfuels/pdfs/hvso_2006/02_mccallen.pdf
8. Nichols DS, Hyams DG, Sreenivas K., Mitchell B, Taylor LK & Whitfield DL (2006) Aerosol Propagation in an Urban Environment. AIAA Paper 2006-3726.
9. Ortega FM & Kambiz S (2004) An Experimental Study of Drag Reduction Devices for a Trailer Underbody and Base. AIAA Paper 2004-2252.
10. Pankajakshan R, Sreenivas K, Mitchell B & Whitfield DL (2007) CFD Simulations of Class8 Trucks. SAE Paper 2007-01-4293.
11. Sreenivas K, Hyams DG, Nichols DS, Mitchell B, Taylor LK, Briley WR, & Whitfield DL (2005) Development of an Unstructured Parallel Flow Solver for Arbitrary Mach Numbers. AIAA Paper 2005-0325.
12. Sreenivas K, Nichols DS, Hyams DG, Mitchell B, Sawyer S, Taylor LK & Whitfield DL (2007) Computational Simulation of Heavy Trucks. AIAA Paper 2007-1087.
13. Storms BL, Satran DR, Heineck JT, & Walker ST (2004) A Study of Reynolds Number Effects and Drag-Reduction Concepts on a Generic Tractor-Trailer. AIAA Paper 2004-2251.

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

David Pointer, Tanju Sofu, Jimmy Chang, and David Weber

Argonne National Laboratory, 9700 S Cass Ave, Argonne, IL, 60439, USA.
David.Pointer@anl.gov

Abstract In preliminary validation studies, computational predictions from the commercial CFD codes Star-CD were compared with detailed velocity, pressure and force balance data from experiments completed in the 7 ft. by 10 ft. wind tunnel at NASA Ames using a Generic Conventional Model (GCM) that is representative of typical current-generation tractor-trailer geometries. Lessons learned from this validation study were then applied to the prediction of aerodynamic drag impacts associated with various changes to the GCM geometry, including the addition of trailer based drag reduction devices and modifications to the radiator and hood configuration. Add-on device studies have focused on ogive boat tails, with initial results indicating that a seven percent reduction in drag coefficient is easily achievable. Radiator and hood reconfiguration studies have focused on changing only the size of the radiator and angle of the hood components without changes to radii of curvature between the radiator grill and hood components. Initial results indicate that such changes lead to only modest changes in drag coefficient.

Introduction

With increasing worldwide concerns about higher air quality standards, greenhouse gas emissions and energy security, significant effort continues to be invested in the development of cleaner, more fuel efficient tractor trailer systems. Modern Class 8 tractor-trailers can have wind average drag coefficients as high as $C_D = 0.6$. At 70 miles per hour, as much of 65% of the total energy available for work that is expended by a typical heavy truck vehicle can be consumed in overcoming aerodynamic drag. Since the energy losses resulting from aerodynamic drag increase as vehicle speed increases, vehicles traveling at higher speeds expend even more energy in overcoming aerodynamic drag forces. A reduction in

the drag coefficient of such vehicles of 50% is not inconceivable and would result in a reduction in fuel use on the order of 25%. [1]

As part of the U.S. Department of Energy Office of FreedomCAR and Vehicle Technologies' Working Group on Aerodynamic Drag of Heavy Vehicles, Argonne National Laboratory is investigating the applicability of commercial Computational Fluid Dynamics (CFD) software for assessment of heavy vehicle aerodynamic characteristics as part of a computational simulation based design process for tractor-trailer systems. This work focuses on the assessment and application of tools and procedures for development and design of strategies or devices for reduction of aerodynamic drag related parasitic energy losses.

Methodology

Prior efforts [2,3,4] as part of the U.S. Department of Energy's Heavy Vehicle Aerodynamics Working Group have evaluated the applicability of commercial CFD software in heavy vehicle design, including assessments of the expected impact of grid resolution and structure on prediction accuracy, the impact of the RANS turbulence model formulation selected, and the impact of considering only half of a vehicle to reduce computational costs. The experience developed by this program provides generic advice for the application of a commercial finite-volume based CFD software package to the prediction of heavy vehicle aerodynamic drag coefficients. In this effort, the experience accumulated in the application of commercial CFD tools to heavy vehicles is applied to enable the quantification of the effects of changing the design of the vehicle. All computational simulations presented herein have used version 3.26 of the commercial CFD code Star-CD. [5]

All computational meshes were generated from CAD data using the automated meshing tools included with Star-CD. In the unstructured hybrid hexahedral/polyhedral computational meshes used in these studies, two separate parameters determine the local size of the mesh. A near-vehicle cell size defines the target cell size in the region immediately surrounding the vehicle surface, and a minimum cell size defines the minimum cell dimension which may be used to resolve curvature of the vehicle surface. Mesh sensitivity analyses included in earlier studies [2,3,4] examined the effects of changes in the near-vehicle cell size and minimum cell size parameters on the prediction of the drag coefficient.

Selection of Tractor Trailer Geometry

All studies presented herein are based on the Generic Conventional Model (GCM), developed by NASA Ames Research Center for scaled wind tunnel testing, is a generalized representation of a conventional aerodynamic U.S. tractor-trailer truck, as shown in Fig. 1. The 1/8th scale model is approximately 2.5 m long by 0.3 m wide by 0.5 m high. The studies contained herein consider experiments that were completed in the NASA Ames 7ft. by 10 ft. wind tunnel. Instrumentation included a force balance, 476 steady pressure transducers, 14 dynamic pressure transducers, and three-dimensional Particle Image Velocimetry (PIV). Data was collected at various Reynolds number values and yaw angles. [6]

Computational Requirements

Models are constructed using a 64-bit 2.8GHz Xeon Linux workstation with 8 GB of RAM. All simulations presented herein utilized the Beowulf cluster maintained by the Nuclear Engineering Division for performing engineering mechanics, fluid dynamics, and reactor engineering analyses. The cluster consists of three front-end (i.e., control) nodes and 100 compute nodes. Each of the 100 compute nodes has a 3.2 GHz Pentium IV processor with 2 GB of memory. The cluster's file server provides nearly 1 Terabyte of home file system space. All of the machines in the cluster are interconnected via Gigabit Ethernet. Typically, 10 computational nodes were used for each simulation.

All cases presented herein could be simulated to steady state convergence – defined by the reduction of the residual for each active equation below 10^{-4} and stabilization of the monitored value of the drag coefficient – with a turn around time of less than one day from the generation of the surface mesh to the completion of the simulation. The optical scan origin of the nominal CAD data makes modification of the geometry more difficult than would be expected for a geometry initially generated by a CAD package. Significant time and effort is required to ensure that all of the complex surfaces created from the optical scan maintain their numerous critical dimensions and connect at each edge to form a single closed volume.



Fig. 1 Generic Conventional Model Geometry

Brief Summary of Prior Validation and Sensitivity Results

Prior studies focused on establishing the validity of commercial CFD software for the prediction of the aerodynamic characteristics of a tractor-trailer vehicle. This previous effort included a mesh sensitivity study considering the effect of near vehicle cell size and minimum cell size on the accuracy of aerodynamic characteristics as well as the impact of turbulence model selection and the use of half vehicle versus full vehicle models. Additional studies have examined the ability to predict the impact of cross-winds on aerodynamic performance. [2,3,4]

Computational Mesh Sensitivity

A series of studies have been completed to evaluate sensitivity of the predictive accuracy to selection of the two primary characteristic dimensions of the generated computational mesh: the near vehicle cell size and the minimum cell size. To assess the impact of the near-vehicle cell size, a series of parametric simulations was completed specifying near-vehicle cell sizes between 6mm and 10 mm for the 1/8th scale GCM. Results, summarized in Table 1, indicate that the drag coefficient can be predicted with acceptable accuracy using a near vehicle cell size of 6 mm. Furthermore, coarser meshes using near vehicle cell sizes as large as 12 mm can likely be used for initial coarse evaluations and trending.

Based on these results, an additional study was completed to evaluate the impact of the minimum cell size. The default minimum cell size in the previous study was 1/8 the near-vehicle cell size. In this study the minimum cell size was reduced to determine whether the same computational accuracy observed in the 6mm case could be achieved using a smaller number of computational cells. When the minimum cell size in the 8 mm case was reduced from 1 mm to 0.5 mm, the accuracy of the drag coefficient prediction improved from 4.2 percent error to 1.0 percent error. This level accuracy was achieved using 30% fewer computa-

Table 1. Effects of Near-Vehicle Cell Size Parameter on Accuracy of Drag Coefficient Prediction.

Near-Vehicle Cell Size (mm)	Predicted Drag Coefficient	Error in Drag Coefficient
experiment	0.398	
16	0.449	12.0
12	0.441	10.3
10	0.418	4.9
8	0.415	4.2
6	0.405	1.7

Table 2. Results of evaluation of two-equation turbulence models for prediction of drag coefficients for the GCM geometry.

Turbulence Model	Predicted Drag Coefficient	Percent Error in Prediction
Experiment	0.398	--
High-Reynolds Number k-epsilon Model	0.402	1.0
Menter k- ω SST model	0.401	0.8
RNG model	0.389	2.3
Chen's model	0.3919	1.61
Quadratic model	0.3815	4.32

tional cells than in the most refined model in the near-vehicle cell size study. Further studies have shown that the further refinement of the near wall cells does not significantly improve the accuracy of the drag coefficient predictions.

Turbulence Model Effects

In all simulations completed for the computational mesh sensitivity studies, the high Reynolds number k- ϵ turbulence model was used in conjunction with a standard logarithmic wall function for the prediction of turbulent kinetic energy and eddy diffusivity. While the high Reynolds number k- ϵ turbulence model is a robust general purpose turbulence model, the strong adverse pressure gradients and large flow recirculation regions associated with the GCM geometry may limit the applicability of generic steady state RANS modeling strategies. Using the computational mesh with a near vehicle cell size of 8 mm and a near wall cell size limit of 0.5 mm, simulations of the aerodynamic characteristics of the GCM model were repeated using five steady RANS turbulence models and their associated wall functions: 1) the standard high-Reynolds number k- ϵ model with logarithmic wall function, 2) the Menter k- ω SST model, 3) the renormalization group (RNG) formulation of the k- ϵ model, 4) the Chen formulation of the k- ϵ model, and 5) the quadratic formulation of the k- ϵ model. Drag coefficients predicted using each of the selected steady-RANS turbulence models are shown in Table 2. The differences in the predicted drag coefficient are largely a result of localized discrepancies in the surface pressure coefficient predictions in the regions of separated flow, with the largest discrepancies appearing in the underbody region just behind the tractor.

Yaw Angle Effects

The effects of cross-winds on the vehicle performance are evaluated by rotating the model in the wind tunnel to introduce an effective yaw angle between the wind direction and the vehicle direction. In the wind tunnel experiments, yaw angles between 14 degrees and -14 degrees were considered. In the experiments, a low drag state is observed at low yaw angles. For yaw angles greater than 3 degrees a higher drag state was observed, where large flow separation zones begin to form along the leeward side of both the tractor and trailer, introducing significant turbulent instability into the flow field.

A series of simulations was completed to evaluate the effects of the computational mesh parameters on predictive accuracy at yaw angles greater than zero. Results of the mesh sensitivity study are summarized in Fig. 2. These studies show that the transition between the high and low drag state can be accurately captured using the methodology developed for the GCM at zero yaw angle. Furthermore, the studies demonstrate that the aerodynamic coefficients of the vehicle can be predicted within a few percent error when sufficient resolution is used near the vehicle surface.

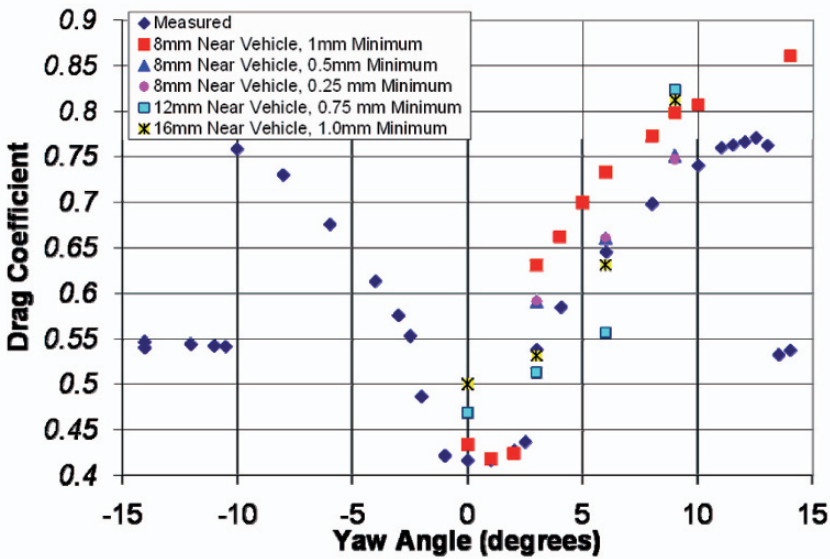


Fig. 2 Comparison of Drag Coefficient Predictions as a Function of Yaw Angle and Mesh Size Parameters.

Ogive Boat Tail Studies

A CAD geometry based on the GCM was developed with a representation of a commercially-marketed inflatable ogive boat tail device attached to the trailer, as shown in Fig. 3. Simulations using this geometry were completed using three different computational mesh configurations developed using the standard process outlined above. Results are summarized in Table 3. When compared with results for the standard configuration of the GCM using comparable mesh densities, a drag coefficient reduction of 7% is expected, which would correspond to a fuel savings of 3.5%. These predictions are consistent with claims based on wind tunnel and road tests by the company that markets the device.



Fig. 3 Modified GCM Geometry with Ogive Boat Tail

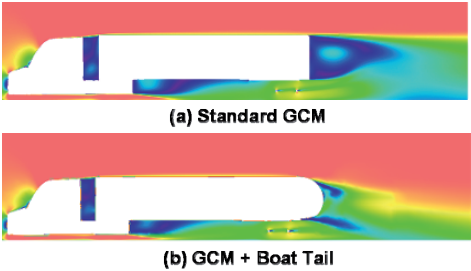


Fig. 4 Velocity Magnitude Predictions for the Standard GCM Geometry and for the Modified GCM Geometry with Ogive Boat Tail

The impact of the device on the wake behind the vehicle is quite dramatic, with the wake closing shortly behind the vehicle rather than many vehicle lengths downstream, as shown in the comparison between results with and without the device in Fig. 4. While this effect should improve driving conditions a vehicle length or more downstream from the rear of the tractor-trailer, as with most trailer boat tail devices, the more energetic turbulence in the reduced-length wake may negatively impact driving conditions for passenger cars near the rear of the vehicle.

Table 3 Summary of Drag Coefficient Predictions for the GCM Geometry with Ogive Boat Tail.

Near Vehicle Cell Size	Near Wall Cell Size	Drag Coefficient
12 mm	2 mm	0.4179
8 mm	1 mm	0.4116
8 mm	0.5 mm	0.3975

Radiator and Grille Modifications

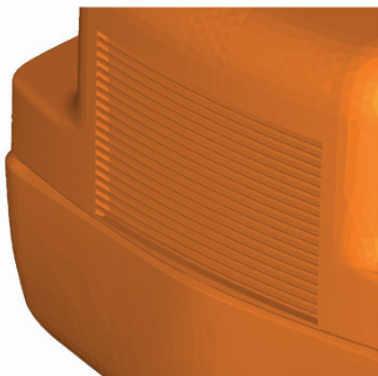


Fig. 5 Grill Geometry of the Modified Generic Conventional Model

In order to meet more restrictive EPA emissions requirements, manufacturers are likely to substantially redesign engine cooling packages to reject more of the engine heat that is currently carried away by the exhaust. An increase in radiator size is likely to result. In order to quantify the potential impact of this change on drag related parasitic energy losses, a series of modified GCM geometries have been developed with modified radiator or grille dimensions. To determine whether the flow of external air through the radiator and into the engine compartment must be modeled when con-

sidering tractor design changes, a series of simulations was completed in which an outlet condition is placed at or near the grill location. In these cases, air entering the engine compartment does not exit to rejoin the bulk flow, as if the vehicle contained a tank of infinite volume into which it was transferred. While not physically representative, this approach allows the effects of flow entering the radiator to be isolated from the effects of flow exiting the underhood region into the bulk flow around the vehicle.

The CAD description of the GCM surface was modified to include an open grill region as shown in Fig. 5. Flow then passes through this grill before exiting the model through an outlet condition placed roughly at the position of the radiator surface. Since flow velocities within the gap between the grill and radiator are very small and the pressure within the gap is much lower than the stagnation pressure, the impact of the inaccuracy of the force calculation at the outlet boundary on the total drag force calculation is expected to be minimal. With the modified geometry, consideration of flow into the radiator results in an increase in the vehicle drag coefficient of approximately 1 percent. Based on these results, it is assumed that the flow into the radiator can be neglected.

Radiator configuration effect studies have considered 4 sets of changes in configuration:

- Changes in grill/radiator height
- Changes in grill/radiator width
- Changes in grill/radiator surface area (aspect ratio maintained)
- Changes in grill/radiator tilt with respect to the vertical axis of the vehicle

In all cases, the GCM geometry is modified such that only the dimensions of the radiator and the angle of the top and side hood panels are changed. Although

the curved segments of the hood region have a complex geometry, the primary radius of curvature between the hood panels and between the top of the hood and the grill is maintained in all cases. A significant number of trial and error iterations between the CAD model and the CFD model were required for each geometric change to ensure that no defects were introduced by the modification of the CAD model which negatively impacted the ability to properly mesh the model or artificially introduced large changes in drag forces. All results presented herein are based on CAD surfaces which have been corrected through this procedure. These issues would almost certainly been avoided had a standard CAD model that was not based on an optical scan been used, since more of the surface construction information would have been recoverable in a standard CAD model.

The effects of changes in grill height on the aerodynamic characteristics of the GCM geometry were evaluated for a 10% reduction in grille height as well as 5% and 10% increases in grille height. To account for the effects of cross-winds on real highway vehicles, yaw angles of 0, 3 and 6 degrees were considered for each case. Predicted drag coefficients are compared with the values obtained for the nominal configuration, as shown in Fig. 6. All changes in drag coefficient observed fall within the expected accuracy of the simulation, which is $\pm 1\%$ for zero yaw and $\pm 5\%$ for higher yaw angles, and no significant impacts on drag-related parasitic losses can be expected to result from these design changes. The conclusion is supported by the surface pressure coefficient contours compared for the three variations and the nominal case in Fig. 7, which show that only the spatial distribution, not the integral sum, of the pressure force acting on the tractor is altered by this change in geometry.

The effects of changes in the width of the radiator and grill region on the aerodynamic characteristics of the GCM geometry were evaluated for both a 10% reduction and a 10% increase in the grill width. Yaw angles of 0, 3 and 6 degrees were again considered for each case. Predicted drag coefficients, shown in Fig. 8, exhibit even less variation that are observed for changes in height and are again within the expected error of the simulations. No significant change in drag force acting on the vehicle is expected as a consequence of small changes in the radiator width. Predicted surface pressure coefficient data, as shown in Fig. 9, again supports this conclusion.

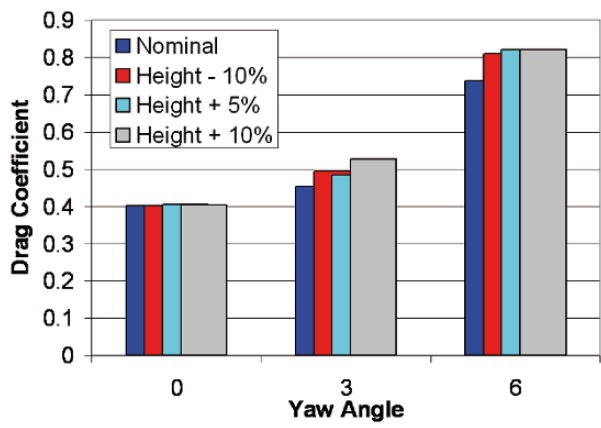


Fig. 6 Predicted Drag Coefficients for Changes in Radiator Height

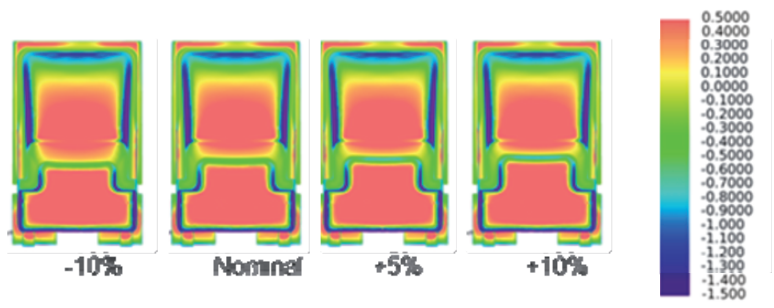


Fig. 7 Predicted Surface Pressure Coefficient Distributions for Changes in Radiator Height

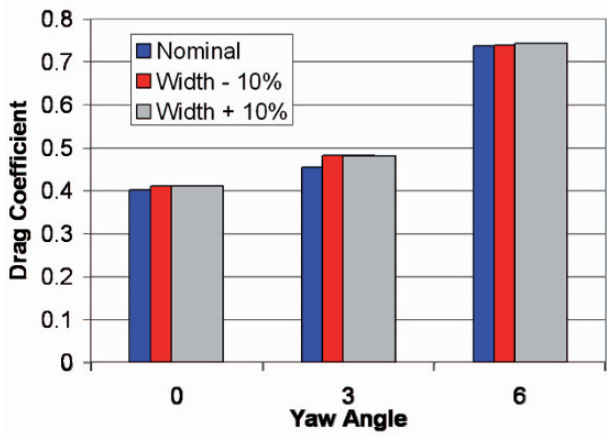


Fig. 8 Predicted Drag Coefficients for Changes in Radiator Width

Since the effects of changes in both height and width of the grill and radiator region were minimal, the effects of changing the total surface area of the radiator and grill region without changing the ratio of the height to the width was evaluated for only a 10% reduction in surface area. Yaw angles of 0, 3, and 6 degrees were again considered to account for cross wind effects. Predicted drag coefficients are shown alongside results for the nominal geometry in Fig. 10. No significant variation in the predicted drag coefficient is observed, and predicted surface pressure coefficient data, shown in Fig. 11, again confirms this conclusion.

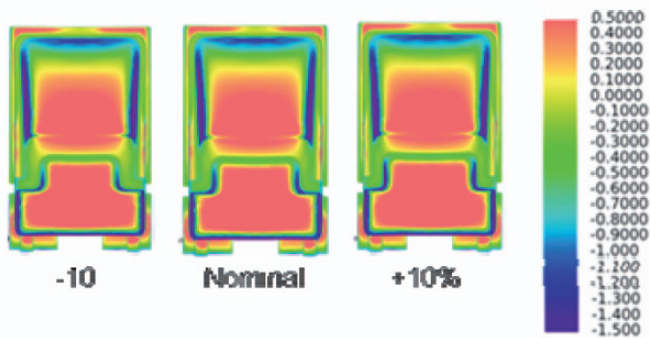


Fig. 9 Surface Pressure Coefficient Distributions for Changes in Radiator Width

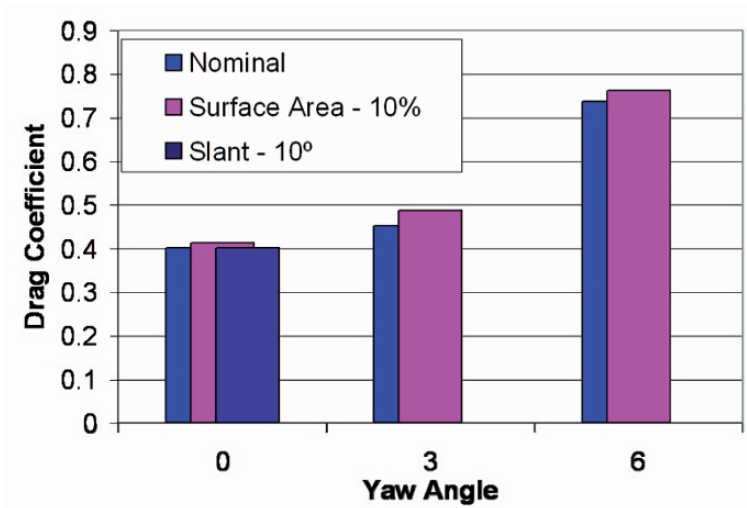


Fig. 10 Predicted Drag Coefficients for Changes in Radiator Surface Area and Radiator Tilt

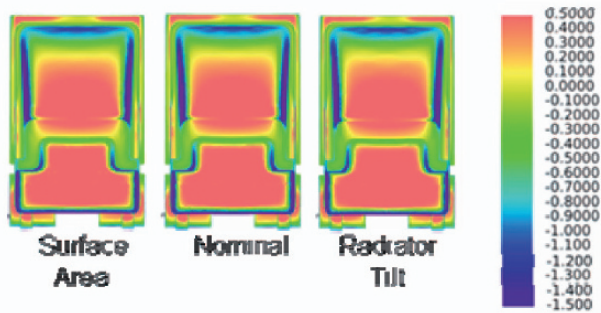


Fig. 11 Surface Pressure Coefficient Distributions for Changes in Radiator Surface Area and Radiator Tilt

In the final study, the effects of the tilt of the grill and radiator are investigated. The bottom of the grill/radiator region is held fixed and the top is tilted backward toward the rear of the vehicle until it is 10° from the vertical axis. Since the drag coefficient of this configuration at zero degrees is with 0.25 percent of the prediction for the nominal geometry, only this case was considered for this configuration. The surface pressure coefficient distribution, also shown in Fig. 11, again confirms this conclusion.

Conclusions

The purpose of this work was to evaluate the applicability of commercial CFD for the quantification of the effects of small and large geometric changes on the aerodynamic performance of a generic tractor-trailer configuration. Studies include the addition of an ogive boat tail to the base of the trailer and small modifications of the tractor's radiator and grille configuration. A nominal geometry based on the Generic Conventional Model was developed and the modified geometries were defined by altering the dimensions of the GCM using commercial CAD software. CFD models were developed using the commercial code Star-CD and the approach developed as part of the authors' prior contributions to the U.S. Department of Energy's Heavy Vehicle Aerodynamic Working Group.

Based on these results, the ogive boat tail concept appears to have the potential to reduce aerodynamic drag related parasitic energy losses and to improve fuel economy. Initial results indicate that small changes in radiator configuration offer neither a significant improvement in fuel economy when reducing the radiator size nor a significant penalty when increasing the radiator size – at least for reasonably aerodynamic vehicles. Small changes in radiator tilt also have a limited effect. These results do not, however, indicate that reductions in radiator size provide no benefit to all tractor trailer vehicles. For less aerodynamic tractors that do not al-

ready take advantage of the very significant benefits of corner rounding with relatively large radii of curvature, reductions in radiator size likely enable the development of new designs which do capitalize those benefits.

Acknowledgement

The submitted manuscript has been created by the University of Chicago as Operator of Argonne National Laboratory ("Argonne") under contract No. W-31-109-ENG-38 with the U.S. Department of Energy.

References

1. McCallen R et al, "DOE's Effort to Reduce Truck Aerodynamic Drag – Joint Experiments and Computations Lead to Smart Design," AIAA Fluid Dynamics Conference, AIAA, Portland, OR, 2004. UCRL-CONF-204819
2. David Pointer, Tanju Sofu, and David Weber, "Development of Guidelines for the Use of Commercial CFD in Tractor-Trailer Aerodynamic Design," 2005 SAE Commercial Vehicle Engineering Conference, Chicago, IL, November 2005. (SAE Paper No. 05CV-120).
3. David Pointer, "Evaluation of Commercial CFD Code Capabilities for Prediction of Heavy Vehicle Drag Coefficients," Proceedings of the 2004 AIAA Fluid Dynamics Conference and Exhibition, Portland, Oregon, June 2004. (AIAA-2004-2254)
4. David Pointer, Tanju Sofu, and David Weber, "Commercial CFD Code Validation for Heavy-Vehicle External Aerodynamics Simulation", in R McCallen, F Browand, and J Ross, eds., *The Aerodynamics of Heavy Vehicles: Trucks, Buses and Trains*, Berlin: Springer Verlag, 2004, pp 473-484.
5. Star-CD, CD-adapco group, Melville, NY.
6. Satran D, "An Experimental Study of the Generic Conventional Model (GCM) in the NASA Ames 7-by-10-Foot Wind Tunnel," United Engineering Foundation Conference on the Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains, United Engineering Foundation, New York, 2002.

Detached Eddy Simulations of a Simplified Tractor/Trailer Geometry

Christopher J. Roy¹ and Harshavardhan A. Ghuge²

¹ Aerospace and Ocean Engineering Department, Virginia Tech, 215 Randolph Hall, Blacksburg, Virginia, 24061-0203, cjroy@vt.edu

² Furnace Improvements, 1600 Highway 6 South, Ste. 480, Sugar Land, TX. 77478, Hghuge@heatflux.com

Abstract Detached Eddy Simulations (DES) are presented for the flow over simplified tractor/trailer geometry at zero degrees yaw angle. The simulations are conducted using a multi-block, structured computational fluid dynamics code. Coarse and fine grids of 3.8 million and 13.2 million cells, respectively, are used for the simulations. The analysis involves estimation of the period of initial flow transients as well as the period required to obtain statistical convergence, and suggests that prior DES studies for this case were not run long enough to ensure statistical independence. Time-averaged quantities compared with experimental data include vehicle drag, surface pressure and wake velocities. The drag coefficient predicted by the DES model matches closely with the experimental value; however, the DES model fails to accurately capture the details of the turbulent flow in the near wake of the trailer base. The simulations on the fine grid show significant improvement over the coarse grid results. Fourier analysis of the unsteady pressures in the base region suggests that the discrepancies in the near wake are/may be due to the improper transfer of turbulence information from the attached boundary layers (which are modeled with the Spalart-Allmaras RANS model) to the turbulent wake where LES-type modeling is used.

Introduction

Tractor-trailers are the prime means for transporting commodities across the US. At a common highway speed of 70 mph, 65% of the total engine output of a Class 8 tractor-trailer is consumed in overcoming the aerodynamic drag [1]. A reduction in the fuel consumption of these tractor-trailers even by a marginal amount can significantly reduce oil consumption and also generate less pollution. One of the ways to reduce the fuel consumption of a tractor-trailer is to reduce the overall aerodynamic drag.

The most common turbulence modeling approach for engineering applications involves solving the Reynolds-Averaged Navier-Stokes (RANS) equations [2]. The RANS equations model the effect of the turbulent eddies on the flow rather than resolving the eddies. Roy et al. [3] have performed RANS simulations on simplified tractor-trailer geometry at a zero degrees yaw angle using the baseline Menter $k-\omega$ model [4] and the Spalart-Allmaras one-equation model [5]. The flow over the front and sides of a tractor-trailer is mostly an attached boundary layer flow; however, there is massive separation in the near base region. Accurate prediction of the pressure in the near base region is important because the pressure in this region plays an important role in the pressure drag acting on the truck. The pressure drag is the major component of the overall aerodynamic drag acting on the tractor-trailer. The prediction of the pressure and velocity in the near base region made by Roy et al. [3] using RANS modeling is not in agreement with the experimental data due to the inability of the RANS models to accurately capture the near-wake three-dimensional vortical structure. Hence RANS modeling cannot be used for capturing the details of a massively separated flow which involves shedding of the vortices. To predict the details of the flow structure in the near wake region the turbulent eddies may need to be resolved. Large Eddy Simulation (LES) is the turbulence modeling technique in which the larger eddies are resolved and the effect of the smaller eddies is modeled using a sub-grid scale model. LES is computationally expensive but is generally more accurate compared to a RANS solution. The biggest drawback of LES is that it is cost prohibitive for modeling wall bounded high Reynolds number flows because of the fine mesh requirements of the model in the boundary layer. To overcome this drawback, the LES model was combined with the Spalart-Allmaras one-equation RANS model [5] to give a hybrid RANS/LES model called the Detached Eddy Simulation (DES) approach. DES was proposed by Spalart et al. [6], and Barone and Roy [7] have evaluated this model for simulation of turbulent wake flow applications. The DES model is used for simulating the flow over simplified tractor/trailer geometry in the current work.

Problem Formulation

A 1/8th scale simplified model of a class-8 tractor-trailer called the Ground Transportation System (GTS) was studied experimentally at the NASA Ames 7×10 ft wind tunnel [8]. Table 1 shows the dimensions of the model. The model is without any wheels and is supported by four posts. There is no tractor-trailer gap in the model, which helps to simplify the grid generation. The Reynolds number based on the trailer width ($W = 0.3238$ m) is approximately 2 million. The origin of the co-ordinate system is in the vertical symmetry plane and at the bottom of the front face of the GTS. The x -axis runs downstream. The y -axis is along the vertical direction and the z -axis runs spanwise towards the side wall. Surface

pressures, wake velocities (from PIV data) and coefficient of drag are used for comparison with the simulations in the present work. Figure 1 shows some of the PIV data planes located near the base of the GTS model which will be used for comparing the u - and v -velocity contours and streamlines. Table 2 shows the conditions of the wind tunnel during the experiment.

Table 1 Dimensions of the GTS

H	0.45 m
L	2.48 m
W	0.32 m

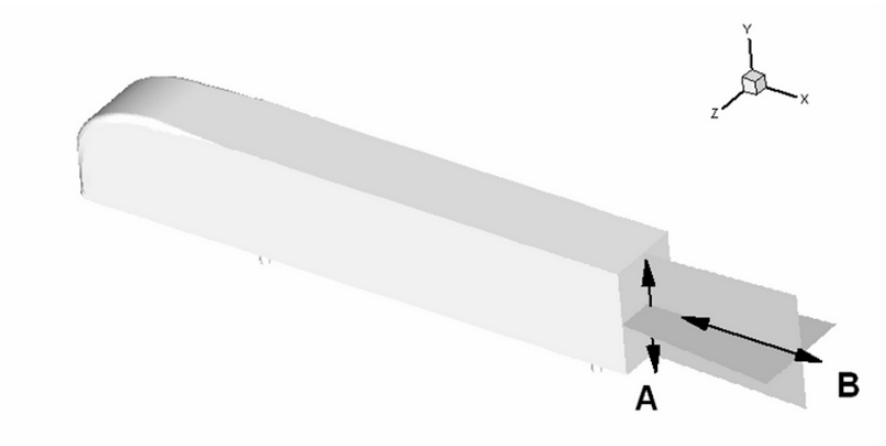


Fig. 1 PIV planes in the near base region

Table 2 Wind tunnel conditions

Parameter	Value
Re	2.02 million
V_∞	93.8 m/s
P_t	102,653.2 N/m ²
P_d	5266.82 N/m ²
T_t	285.76 K
	1.18 kg/m ³
P_{ref}	97,216. N/m ²
M_{ref}	0.279

Simulation Approach

The code used for the simulations was the Sandia Advanced Code for Compressible Aerothermodynamics Research and Analysis (SACCARA). It is a compressible flow solver used at Sandia National Laboratories and was originally developed from the INCA code [9][10] written by Amtec Engineering. It is used to solve the Navier-Stokes equations and the turbulence transport equations in finite-volume form. Code verification has been performed on the SACCARA code by code-to-code comparison with other Navier-Stokes codes [10] and with Direct Simulation Monte Carlo method [11]. The code uses a Lower-Upper Symmetric Gauss-Seidel scheme to solve the discretized equations. This scheme is based on the works of Yoon et al. [12] and Peery and Imlay [13] and has excellent scalability up to thousands of processors.

The SACCARA code uses a cell-centered finite volume approach. The code uses Yee's Symmetric Total Variation Diminishing (STVD) upwind scheme [14] for discretization of the convective terms in the Navier-Stokes equations. The scheme is modified to incorporate Harten's artificial compression switch (ACM) [15] to further reduce the dissipation, along with characteristic based filtering [14]. This modified STVD scheme uses the constant K to control the global numerical dissipation in the solution. The value of K varies from 0.03 to 2 and is problem dependent. For the current simulations, we employed $K = 0.2$. Harten's switch is used to reduce the numerical dissipation locally. The viscous terms were discretized using central differencing. A sub-iterative procedure is used to obtain second-order accuracy during the temporal discretization, where the summation of the discretized temporal derivative and the steady-state residual at the $n + 1$ time level is iterated until it reaches the specified tolerance value. Unless otherwise specified, for the coarse mesh the number of sub-iterations performed during each time step were $n_{it} = 4$, and for the fine mesh the number of sub-iterations performed during each time step were $n_{it} = 5$.

The destruction term in the Spalart-Allmaras one-equation model, when balanced with its production term, adjusts the eddy viscosity $\tilde{\nu}$ to scale it with the local strain rate S and the distance from the wall d , i.e., $\tilde{\nu} \propto Sd^2$ [6]. The Smagorinsky eddy viscosity model scales its subgrid scale eddy viscosity with S and grid spacing Δ i.e., $\tilde{\nu} \propto S\Delta^2$ [6]. Spalart modified the Spalart-Allmaras one-equation model in which the distance from the wall d is replaced by $\tilde{d}$

$$\tilde{d} = \min(d, C_{DES}\Delta)$$

where Δ is maximum mesh spacing in the three coordinate directions

$$\Delta = \max(\Delta x, \Delta y, \Delta z)$$

and C_{DES} is the constant proposed by Spalart and is equal to 0.65 [6]. Away from the wall, $d > \Delta$ i.e., $\tilde{d}$ becomes equal to $C_{DES}\Delta$ and the model is modified to give Smagorinsky's subgrid LES model. Near the wall $d < \Delta$ and the model behaves as the Spalart-Allmaras one-equation model. The DES model is thus a combination of the Spalart-Allmaras one equation model and Smagorinsky's subgrid-scale LES model.

The simulations were performed over a domain that included a significant portion of the wind tunnel test section with the GTS model insertion location consistent with the experiment. A coarse mesh having 3.8 million cells and a fine mesh having 13.2 million cells were used to perform the study. Figure 2 shows the coarse mesh around the base of the GTS in the vertical symmetry plane. The grids were generated such that the highest value of the wall y^+ term is 1.4 from the coarse mesh. The structured mesh employed used point to point connectivity at the interface of two blocks. Both the grids were decomposed and solved using 86 processors.

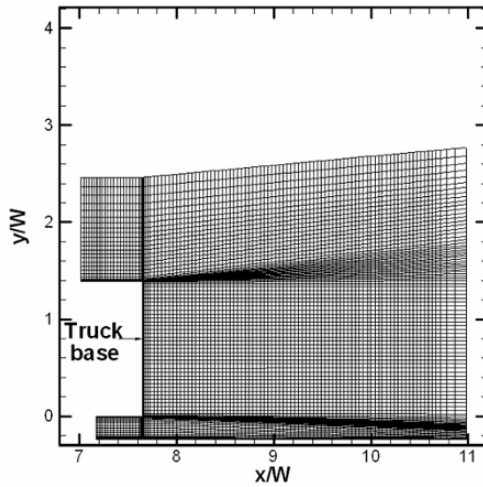


Fig. 2 Coarse mesh near the base of the GTS ($z/W = 0$)

The inflow boundary employs stagnation values for pressure $P_t = 102,653 \text{ N/m}^2$ and temperature $T_t = 285.76 \text{ K}$ and enforces inflow normal to the boundary. The outflow boundary used a fixed static pressure of $P = 97,700 \text{ N/m}^2$. The back pressure was chosen so that the tunnel wall reference pressure (P_{ref}) located at $x/W = 4.47$, $y/W = 2.588$ and $z/W = -4.7$ matched with the experiment. The reference pressure is used in the calculation of the pressure coefficient and drag coefficient.

$$C_p = \frac{P - P_{ref}}{P_t - P_{ref}} \quad \text{and} \quad C_D = \frac{D}{(P_t - P_{ref})(WH)}$$

Table 3 compares average reference pressure values from the coarse and fine mesh with the experimental value. Slip wall conditions were applied to the top and side walls of the wind tunnel to save the computational cost, while the floor of the wind tunnel, the GTS surface and the support posts employ no-slip velocity conditions and assume an adiabatic wall. The freestream dynamic eddy viscosity is set at $\mu_t = 1 \times 10^{-5} \text{ N/m}^2$. Solid wall boundary condition for the turbulence model can be found in [16].

Table 3 Comparison of reference pressure (P_{ref}) from the experiment and time-averaged reference pressure (P_{ref}) from simulations

Experiment	Coarse Mesh	Fine Mesh
97,336 N/m ²	97,295 N/m ²	97,249 N/m ²

Time has been non-dimensionalized by a reference time scale τ defined as $\tau = W / V_\infty = 0.0034s$

where V_∞ , the freestream velocity is equal to 93.8 m/s. The coarse mesh simulations were performed using a non-dimensional time step $\Delta\tau_s = 1.16 \times 10^{-3}$ ($\Delta t_s = 4 \times 10^{-6} \text{ s}$) and the fine mesh simulations were performed using a non-dimensional time step of $\Delta\tau_s = 2.32 \times 10^{-3}$ (i.e., $\Delta t_s = 8 \times 10^{-6} \text{ s}$). Table 4 gives the characteristic length scales for the fine and coarse meshes (note that the length scales are non-dimensionalized by the width of the trailer).

Table 4 Characteristic length scales

Mesh	Δ_{wall}	Δ_{wake}	Points across the wake (considering maximum grid spacing)
Coarse	6.17×10^{-5}	0.038	26
Fine	3.08×10^{-5}	0.026	38

Numerical Accuracy

DES is a simulation of an unsteady flow i.e., it yields a time-accurate solution. To assess the mean properties of the flow the data from the simulations have to be averaged over some time window. The size of the time window is determined by performing a statistical convergence analysis. The statistical convergence analysis performed during this research involved averaging the data from the unsteady solution over different time windows. The time windows used for averaging the data are shown in Fig. 3. This figure shows pressure signal from the simulations located in the base of the GTS ($x/W = 0.64$, $y/W = 0.63$ and $z/W = -0.46$). The signal is plotted against non-dimensional time τ . It is assumed that the largest time window shown in the figure is wide enough to give statistically converged results. Also, computation of a bigger time window will increase the computational cost. The

width of this window is $\Delta\tau_w = 270$ ($\Delta t_w = 0.91$ s). Time-averaged velocity profiles corresponding to the time windows shown in Fig. 3 were used for this analysis. The line B shown in Fig. 1 is the location where the velocity profiles were computed respectively. These locations are in the turbulent wake of the flow over the GTS. Figure 4 compares the differences between the various u/V_{ref} velocity profiles corresponding to the smaller time windows shown in Fig. 3 and the u/V_{ref} velocity profile corresponding to the biggest time window. The velocity profiles corresponding to time windows having starting time $\tau_{start} = 394$ and beyond have minimal difference with respect to the corresponding profile from the biggest window. An upper and lower limit is placed on the results from the smaller time windows. These limits are equal to the maximum difference between the variable calculated from the corresponding smaller time window and the variable obtained from biggest time window/best available solution. This analysis suggested that the smallest possible time window giving results close to the statistically converged values is the window corresponding to $\tau_{start} = 394$, $\tau_{end} = 571$ (i.e., $\Delta\tau_w = 177$ or $\Delta t_w \approx 0.612$ s). Any window which is smaller than this is not wide enough to give results close to the converged values. Hence all the comparison with the experimental data is made by time averaging the data from the simulations over a period of $\Delta t_w = 0.612$ s. Unaune et al. [17] and Maddox et al. [18] have reported the statistical convergence period of $\Delta t_w \approx 0.35$ s and 0.27 s, respectively.

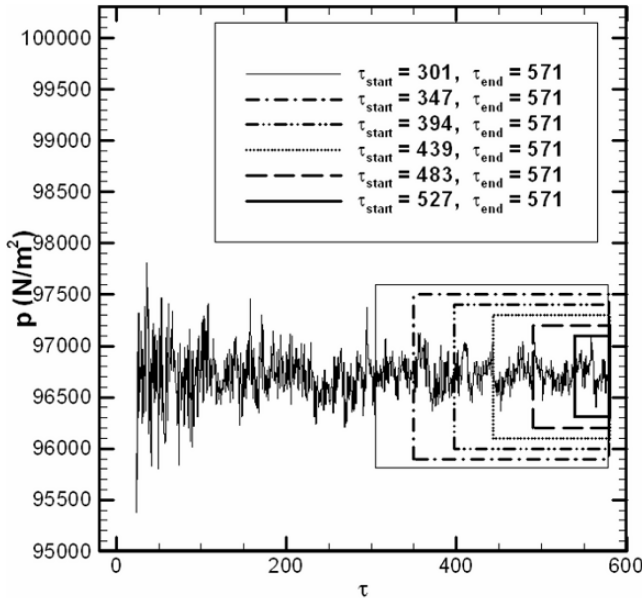


Fig. 3 Time windows for statistical convergence

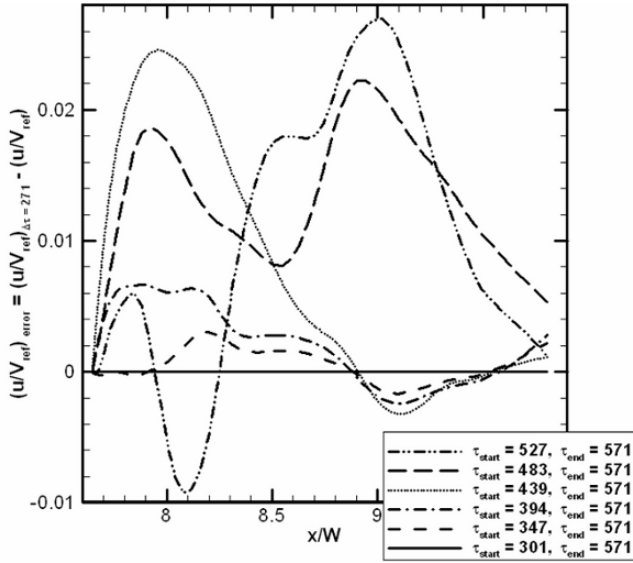


Fig. 4. Error in axial velocity relative to largest window: coarse mesh ($K = 0.2$)

The simulations are initialized with freestream values of velocity, pressure and temperature. The flow over the GTS during some initial time period oscillates because of the waves generated due to sudden exposure of the GTS to the freestream flow. These oscillations are called the initial transients. The large pressure oscillations seen in Fig. 5 near $\tau = 0$ are the initial transients in the flow. As the flow over the GTS settles the initial transients diminish. The data collected during the initial transient period should be neglected. Therefore the time period of existence of initial transients has to be calculated. For calculating the period of initial transients the pressure profiles along the line A shown in Fig. 1 was time averaged over a window with $\Delta\tau_w = 177$. $\Delta\tau_w = 177$ is the time window required to achieve statistically converged results which was determined in the previous section. The time window was slid along the time axis as shown in Fig. 5. Figure 6 compares the differences between C_p profiles corresponding to the various earlier time windows shown in Fig. 5 and the C_p profile from the window farthest along the time axis. It is assumed that the window farthest along the time axis is far enough from $\tau = 0$ and does not involve any initial transients. Pressure and velocity profiles corresponding to time windows having starting time $\tau_{\text{start}} = 301$ and beyond have minimal difference with respect to the corresponding profiles from the farthest window. This suggests the existence of initial transients up to $\tau = 301$, i.e., $t = 0.92$ s. The data collected after this period has marginal effect of the initial transients and can be used for comparison with the experiment. This will be done by considering an upper and lower limit over the time-averaged data from the simulations denoting the effect of the initial transients. Initial transience period

predicted by Unaune et al. [17] is 0.2 s which is significantly less than that predicted here.

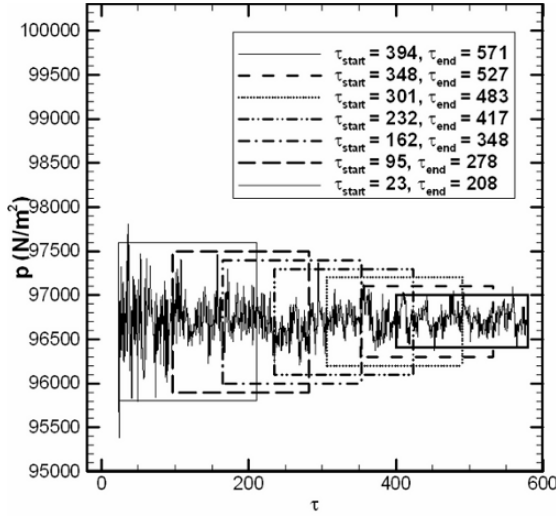


Fig. 5 Time windows for estimating initial transients

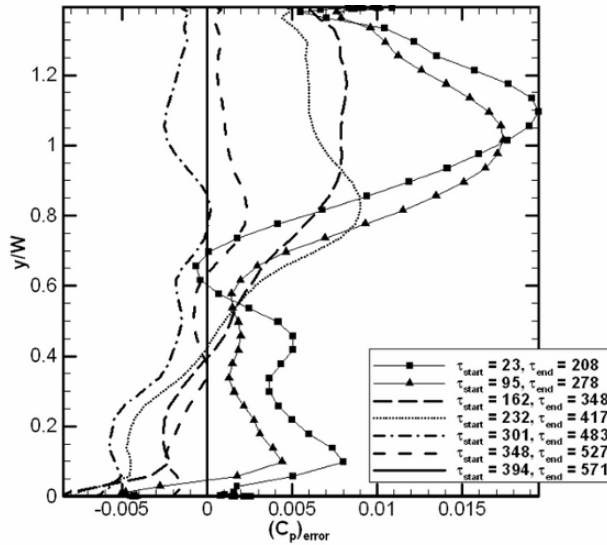


Fig. 6 Cp profile error relative to final time window: coarse mesh ($K = 0.2$)

The maximum uncertainty due to the effect of insufficient statistical convergence, presence of initial transients, temporal discretization (see Ref. [19]) and

insufficient iterative convergence (see Ref. [19]) was calculated for the pressure and velocity of the flow. Table 5 gives the values of the uncertainty from these sources. The total uncertainty was calculated using an analogy to experimental uncertainty propagation techniques using the formula

$$U_T = \sqrt{U_1^2 + U_2^2 + U_3^2 + U_4^2}.$$

Table 5 Maximum uncertainty bounds

Source of uncertainty	C_p	u/V_{ref}	v/V_{ref}	w/V_{ref}
Initial transients (U_1)	0.006	0.01557	0.01524	0.0212
Statistical convergence (U_2)	0.0047	0.00665	0.01103	0.01396
Number of sub-iterations (U_3)	0.007	0.01915	0.015	0.02296
Time step (U_4)	0.0175	0.05116	0.0430	0.00564
Total uncertainty (U_T)	0.028	0.0572	0.0493	0.0346

The effect of mesh size is studied by comparing results from the coarse and the fine meshes. Figure 7 compares the u/V_{ref} velocity contours and streamlines from the coarse and fine mesh in the vertical streamwise plane (shown in Fig. 1). Figure 7 (top) shows that the coarse mesh simulations predict the upper vortex dominant over the lower vortex. The fine mesh results shown in Fig. 7 (bottom) show the vortices are almost equal in size. Figure 8 compares the pressure coefficient along the line A in Fig. 1. The coarse mesh pressure profile shows a bulge in the negative direction in the upper half of the trailer base due to the dominant upper vortex in the coarse mesh. The fine mesh pressure profile is almost a flat curve due to equal size of the upper and lower vortices (Fig. 7, bottom). It should be noted that the $\Delta\tau_s$ and n_{It} values used for the fine and coarse mesh simulations are different. However the effect of $\Delta\tau_s$ and n_{It} on the results from the simulations is relatively small when compared to the effect of the mesh size. The fine mesh results, which are expected to be more accurate, are now compared with the experiment.

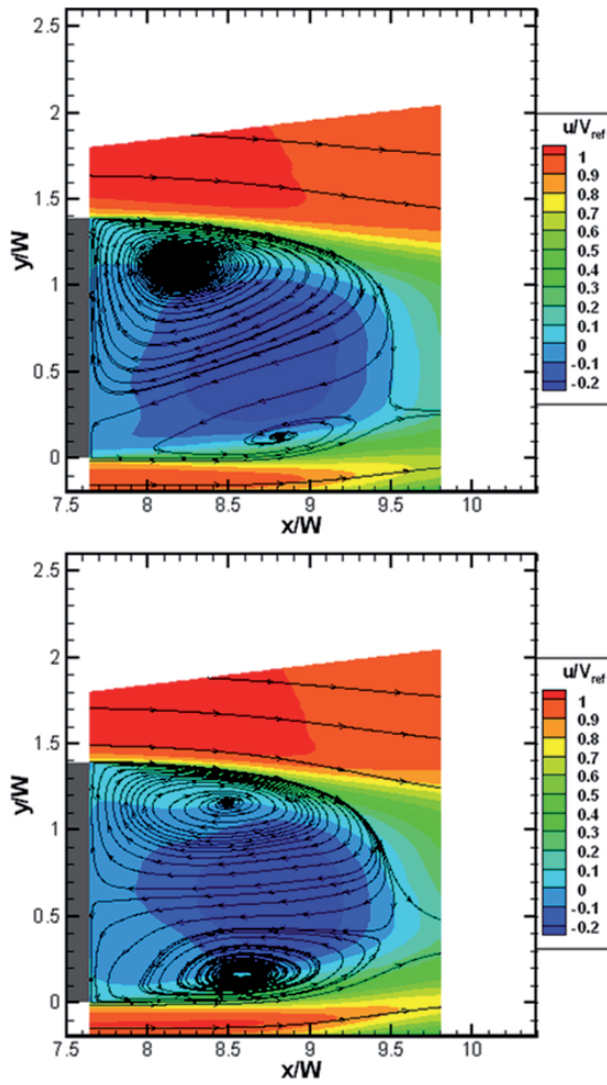


Fig. 7 Axial velocity contours and streamlines for the coarse mesh (top) and fine mesh (bottom)

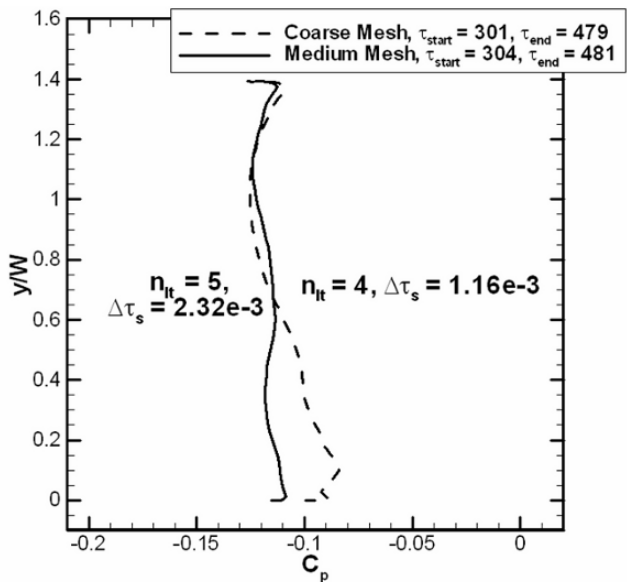


Fig. 8 Effect of mesh size on the base pressure distribution

Results and Discussion

Figures 9a and 9b compare the u/V_{ref} velocity contours and streamlines in the vertical streamwise plane (vertical plane in Fig. 1) from the experiment and fine mesh simulations. The experimental data shows the lower vortex located much closer to the base of the GTS model and at a higher location than the simulations. The upper vortex is not seen in the experimental PIV window but the nature of the streamlines in the window suggests that it may be located near the upper right corner of the window. The distribution of the vortices in the fine mesh simulations is more symmetric as seen in Fig. 9b. Figure 9c shows the u/V_{ref} velocity contours and streamlines in the vertical streamwise plane from the RANS modeling performed by Roy et al. [3]. The figure shows very similar structure of the upper and lower vortices when compared to the DES simulations. However the vortices from the RANS modeling are closer to the base of the GTS model than the results from DES simulations. This difference in location of the vortices from the experiment and the simulations gives a major difference in the pressure distribution on the base of the GTS model obtained from the experiment and the simulations (to be discussed later).

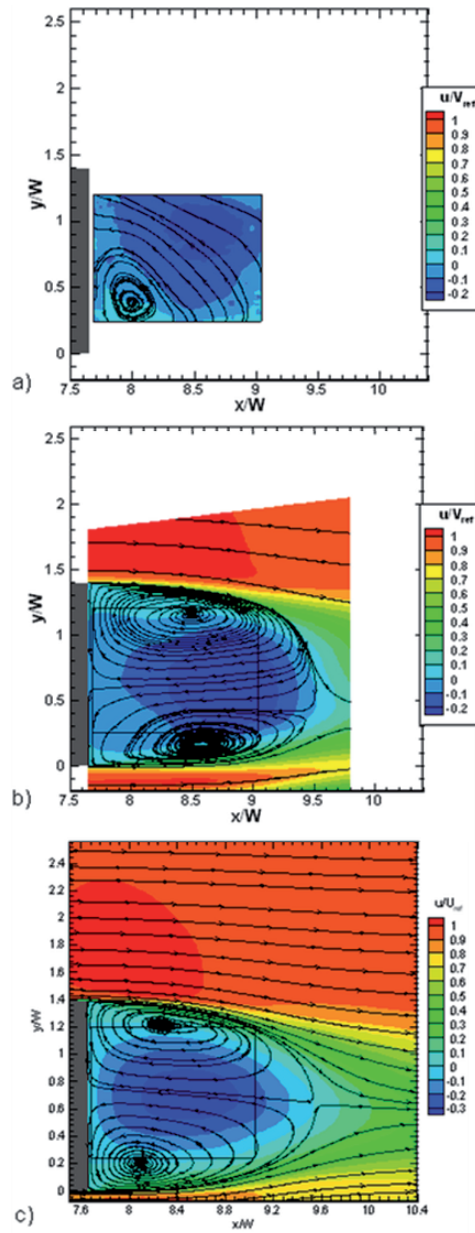


Fig. 9 Axial velocity contours and streamlines for a) experiment [8], b) fine mesh DES, and c) RANS computations [3]

Figures 10a, b and c compare the u/V_{ref} , v/V_{ref} and w/V_{ref} velocity profiles, respectively, from the experiment and the simulations. These profiles are plotted along the line B shown in the base of the GTS model in Fig. 1. The uncertainty bounds on the profiles from the simulations are obtained from Table 5. Out of the two PIV data sets one was obtained from the vertical streamwise PIV data plane (vertical plane in Fig. 1) and the other data set is obtained from horizontal streamwise PIV data plane at $y/W = 0.696$ (horizontal plane in Fig. 1). The difference in the experimental data sets may be the result of errors in the PIV measurements. Figure 10a shows that the recovery of the u -velocity from the experiment starts earlier than the simulations. Figure 10b shows the v -velocity profiles from the experiment has a completely different trend than the profile from the simulations. Figure 10c shows a comparison of the w -velocity. Since these profiles are in the $z/W = 0$ plane i.e., the vertical symmetry plane of the GTS the w -velocity should ideally be zero in this plane and we see the same in the profiles from the simulations and the experiment.

Figure 11a compares the pressure distribution plotted along the line A in the base of the GTS as shown in Fig. 1. It should be noted that in Fig. 11a the pressure profiles are plotted on a magnified C_p scale, so that even though it appears that the profile from the DES simulations and the profile from the RANS modeling performed by Roy et al. [3] do not match with the experimental data, the overall range of the pressure predicted by the computational models match with the range of the experimental pressure values. The plot shows that the profile from the DES simulations is almost a flat curve but the RANS model shows the opposite trend of pressure distribution to that seen in the experiment. The uncertainty bound on the profile from DES simulations is obtained from Table 5. Figure 11b compares the pressure distribution along the top and bottom surfaces of the GTS in the $z/W = 0$ plane from the experiment and the simulations. The top and bottom surfaces are in the attached boundary layer flow region where the DES model does a good job of the matching with the experimental data.

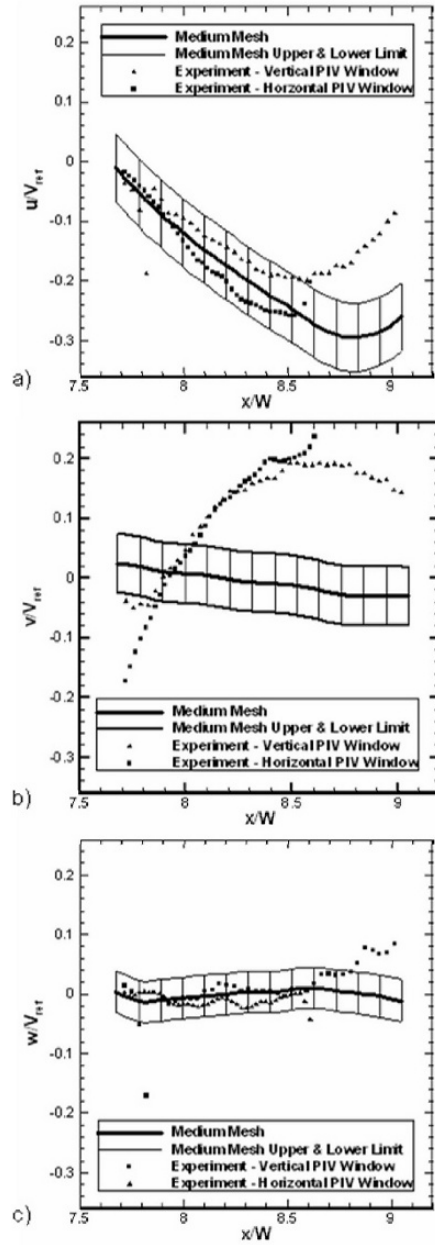


Fig. 10 Comparison of velocity profiles from experiment and DES simulations along the axial line in the wake a) u/V_{ref} , b) v/V_{ref} and c) w/V_{ref}

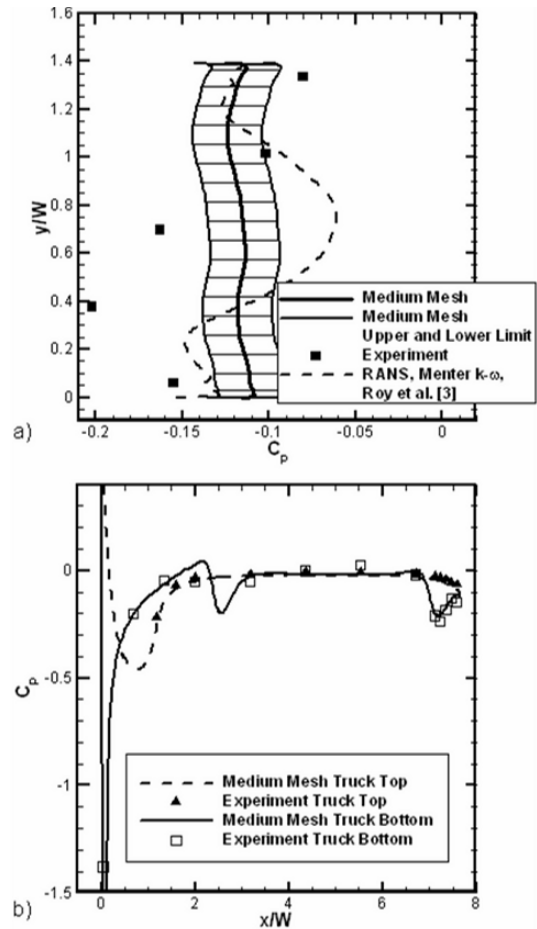


Fig. 11 Pressure coefficient profiles for a) the trailer base and b) the top and bottom of the GTS

Table 6 compares the drag coefficient from the simulations with the experimental data. The table shows that the DES simulations of the fine mesh, coarse mesh and the DES by Unaune et al. [17] do a very good job of predicting the drag. The reason for the high value of error in the drag predicted by Maddox et al. [18] is not known. RANS models overpredict the drag value by a large amount as compared to the DES simulations.

Table 6 Drag coefficient

Source	C_D	% error w.r.t. experiment
Experiment	0.249	0%
Coarse mesh DES (3.8M cells)	0.2565	+3 %
Fine mesh DES (13.2M cells)	0.2439	-2 %
Unaune et al. [17]DES (13.8M cells)	0.253	+1.6%
Maddox et al. [18]DES (6M cells)	0.279	+12%
Roy et al. RANS Menter k- ω (20M cells)	0.298	-19%
Roy et al. RANS Spalart-Allmaras (20M cells)	0.413	+65%

In the experiment a high-frequency pressure transducer is located on the base of the GTS model ($x/W = 7.647$, $y/W = 0.63$ and $z/W = -0.46$). The power spectra for the unsteady pressure signal is obtained by performing a fast Fourier transformation (FFT) of the normalized unsteady pressure signal from the experiment and the simulations. FFT is an efficient way to perform a discrete Fourier transform of a signal. It converts the signal from the time domain into the frequency domain. This power spectrum serves the purpose of giving an idea about the frequency of the turbulent structures in the region near the base of the GTS model. The frequency corresponding to the peak in the power spectra is the frequency containing the most turbulent energy. Although not shown, the power spectra obtained from the experiment shows a peak frequency at nearly 24Hz. The power spectra from the simulations show three peaks at approximately 80 Hz, 110 Hz and 130 Hz. Also some peaks are seen in the range of 10 to 30 Hz which appear to roughly correspond to the dominant frequency found in the experiment. Animations of the turbulent flow in the wake were performed and they showed the flapping of the shear layers which separate from the attached boundary layers on the top and bottom surfaces of the GTS in the LES region in the wake. The peaks in the simulation power spectra appear to correspond to the large-scale unsteadiness of the shear layers observed in the animations. We surmise that the presence of the additional peaks in the power spectra from the DES simulations (which are not found in the experiment) is due to the inability of the RANS boundary layer to properly transfer turbulence fluctuation information into the shear layer in the LES region. This is an acknowledged short-coming of the DES method and may in fact be responsible for the inability of the DES method to reproduce the wake structure seen in the experiments.

Conclusions

The estimated initial transient period ($t \approx 1$ s) and the time period required for achieving statistical convergence ($t \approx 0.61$ s) should provide guidance for future simulations. The drag value predicted by the simulations agrees with the experimental value. The pressure values predicted by the DES in the attached boundary layer regions also matches with the experiment. The range of the pressure values predicted by the DES simulations in the base of the GTS model agrees with the range of the values of pressure from the experiment. The actual pressure profile in the base region, however, does not match with the experimental data. The pressure profile from the DES simulations however gives much better prediction than RANS modeling [3]. The prediction of the location of the vortices near the base of the model does not match with the experimental data. Here the DES model fails to accurately capture the details of the turbulent flow in the near base region. The failure of the current DES simulations to predict wake details appear to be related to one of the main drawbacks of the hybrid RANS/LES methods, namely that there is no mechanism to transfer turbulence fluctuation information from the RANS region (i.e., the boundary layer) to the LES region (i.e., the wake). Hence when the shear layers separated from the surface of the trailer enter the wake region governed by the LES model, there is no way to convert the large, modeled eddy viscosity into large-scale turbulent fluctuations required for the LES model. Research on the transfer of information from the modeled turbulence obtained from RANS equations in the boundary layer to the LES model in the separated region should improve the predictions of the wake flow details given by DES on coarser grids.

References

- [1] MacCallen R, Salari K, Ortega J, Browand F, Hammache M, Hsu T, Leonard A, Chatelain P, Rubel M, Roy C, DeChant L, Hassan B, Heineck J, Yaste D, Storms B, Englar R and Pointer D, 2004, "DOE's Effort to Reduce Truck Aerodynamic Drag - Joint Experiments and Computations Lead to Smart Design," 34th AIAA Fluid Dynamics Conference, June 28 - July 1, Portland, OR.
- [2] Wilcox D, 1998, *Turbulence Modeling for CFD*, 2nd Ed., DCW Industries, Inc., La Canada, CA.
- [3] Roy C, Payne J and Payne M, 2006 "RANS Simulations of a Simplified Tractor/Trailer Geometry," *Journal of Fluids Engineering*, Vol. 128, pp. 1083-1089.
- [4] Menter F, 1994, "Two-equation Eddy-viscosity Turbulence Models for Engineering Applications" *AIAA Journal*, Vol. 32, pp. 269-289.

- [5] Spalart P and Allmaras S, 1994, "A One-Equation Turbulence Model for Aerodynamic Flows," *La Recherche Aerospatiale*, 1, pp. 5-21.
- [6] Spalart P, Jou W, Strelets M and Allmaras S, 1997, "Comments on the Feasibility of LES for Wings and on a Hybrid RANS/LES Approach," *Advances in DNS/LES*, Columbus, OH, Greyden Press.
- [7] Roy C, DeChant L, Payne J and Blottner F, 2003 "Bluff-Body Flow simulations Using Hybrid RANS/LES," *AIAA 2003-3889*.
- [8] Storms B, Ross J, Heineck J, Walker S, Driver D and Zilliac G, 2001, "An Experimental Study of the Ground Transportation System (GTS) Model in the NASA Ames 7- by 10-Ft Wind Tunnel," *NASA/TM-2001-209621*.
- [9] INCA User's Manual, Version 2.0, 1995, Amtec Engineering, Inc., Bellevue, WA.
- [10] Payne J and Walker M, 1995 "Verification of Computational Aerodynamic Predictions for Complex Hypersonic Vehicles using the INCA Code," *AIAA Paper 95-0762*.
- [11] Roy C, McWherter-Payne M and Oberkampf W, 2003, "Verification and Validation for Laminar Hypersonic Flowfields Part 1: Verification," *AIAA Journal*, 41 (10), pp. 1934-1943.
- [12] Yoon S and Jameson A, 1987 "An LU-SSOR Scheme for the Euler and Navier-Stokes Equations," *AIAA Paper 87-0600*.
- [13] Peery K and Imlay S, 1986 "An Efficient Implicit Method for Solving Viscous Multi-Stream Nozzle/Afterbody Flow Fields," *AIAA Paper 86-1380*.
- [14] Yee H, Sandham N and Djomehri M, 1998 "Low-dissipative High-Order Shock-Capturing Methods Using Characteristic-Based Filters," *Journal of Computational Physics*, Vol. 150, pp. 199-238.
- [15] Harten A, 1984 "On Class of High Resolution Total-Variation-Stable Schemes," *SIAM Journal of Numerical Analysis*, Vol. 21, No. 1, pp. 1-23.
- [16] Roy C and Blottner F, 2003 "Methodology for Turbulence Model Validation: Application to Hypersonic Flows," *Journal of Spacecrafts and Rockets*, Vol. 40, No. 3. pp. 313-325.
- [17] Unaune S, Sovani S and Kim S, 2005, "Aerodynamics of a Generic Ground Transportation System: Detached Eddy Simulations," *SAE Technical Papers*, 2005-01-0548.
- [18] Maddox S, Squires K, Wurtzler K and Forsythe J, 2004, "Detached-Eddy Simulation of the Ground Transportation System," *The Aerodynamics of Heavy Vehicles: Trucks, Buses and Trains*, R McCallen, F Browand, and J Ross, Eds., Springer, Berlin, Germany, pp. 89-106.
- [19] Ghuge H, "Detached Eddy Simulations of a Simplified Tractor-Trailer Geometry," *Masters Thesis*, Auburn University, May 2007.

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

Kambiz Salari, Paul Castellucci

Lawrence Livermore National Lab, USA
salari1@llnl.gov

Abstract Currently, there exists a lack of confidence in the computational simulation of turbulent separated flows at large Reynolds numbers. The most accurate methods available are too computationally costly for use in engineering applications. Using concepts borrowed from large-eddy simulation (LES), a two-equation Reynolds-averaged Navier-Stokes (RANS) turbulence model is modified to simulate the turbulent wakes behind bluff bodies. This modification involves the computation of one additional scalar field, adding very little to the overall computational cost. When properly inserted in the baseline RANS model, this modification mimics LES in the separated wake, yet reverts to the unmodified form near no-slip surfaces. In this manner, superior predictive capability may be achieved without the additional cost of fine spatial resolution associated with LES near solid boundaries. Simulations using several modified and baseline RANS models are benchmarked against both LES and experimental data for a circular cylinder wake at Reynolds number 3900. These results reveal substantial improvements using the modified system and appear to drive the baseline wake solution toward that of LES, as intended. Further results include the simulation of the turbulent wake created by the Ground Transportation System (GTS), a simplified tractor-trailer geometry studied extensively by the Department of Energy for use in turbulence model validation.

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

Simon Watkins

School of Aerospace, Mechanical and Manufacturing Engineering, RMIT University,
Melbourne, Australia
simon@rmit.edu.au

Abstract An accurate and repeatable measurement of truck-induced spray is required to develop and test spray-reducing devices. Such a system is described, based on a sequence of CCD-captured video images of a black and white chequer-board which was partially obscured by spray from a passing test truck. Images were analyzed to reveal the contrast changes, from which spray densities were inferred. Results of on-road trials are described and it was found that none of the tested wheel-mounted systems offered any statistically significant reduction in spray. Supporting track and wind-tunnel tests documented the flow vectors close to the truck; for an unmodified vehicle and when fitted with sideskirts and a cab-mounted add-on aerodynamic device. It was concluded from the flow field studies that the problem of vision impairment for a passing motorist would be significantly reduced when sideskirts and (for vehicles hauling a high load) correctly-matched cab roof deflectors were fitted. This offers the trucking industry the combined advantages of drag and spray reduction.

1. Introduction

Trucks traveling on wet roads are known to present a nuisance to other road users; particularly when attempting overtaking maneuvers. Whilst some contention exists over the extent of danger this poses, it is clearly of annoyance to other road users and known to be a source of cleaning costs to the trucking industry [1].

There have been many attempts to retrofit several types of commercial vehicles, including buses, with devices designed to ameliorate the spray via inserts or other changes or treatment to the mudguards. These have included fabric materials, bristles and specially shaped inserts which all try to catch splash and spray in order to avoid the vision impairment associated with lateral passage of spray into the area of passing vehicles. However, analyzing the effectiveness of these devices

has proved difficult and unreliable and in 2000 a report to the USA Congress [2] concluded that there was “significant interest in splash and spray suppression by government and the private sector”. It also found that “technologies that purport to reduce splash and spray cannot be shown to translate into real world visibility improvements for drivers” and that when reviewing work in Europe “there are no data that show devices that are approved under their laboratory test result in better visibility on vehicles on the road”.

There have been several experiments conducted in Australia which have provided insights into methods to repeatably document vision impairment and to quantify the influences of mudguard treatment and the influence of aerodynamics on the movement of the spray. Other attempts to counter the lack of a reliable method and to investigate the influence of devices include [3, 4, 5, 6]. These have generally involved the use of video acquisition and analysis techniques to quantify truck-induced spray, although other laser-based methods have been used, prior to the advent of relatively low cost PCs and CCD cameras.

The video-based spray-measuring system described herein was developed at RMIT University, on behalf of the Australian Federal Office of Road Safety. From the conception of the project it was decided that the system should represent what an overtaking motorist would experience as closely as possible. This system and some typical results are described here with an overview of some supporting flow field studies.

2. Spray-Measuring System

The spray-measuring system uses the change in contrast of recorded video images to determine the density of spray. Whilst it was considered that the measurement of brightness could have a use in the measurements, a series of laboratory tests, where spray was generated in the laboratory and videoed under a range of different conditions [7] demonstrated the undesirable influence of brightness on the results. Clearly for any road or track tests this would prove problematic since the changing light (including the influence of clouds shading direct sunlight) would be an unwanted variable. The choice of contrast as the defining variable was based on the fact that objects, such as other motor vehicles, are seen by the human eye due to their contrast with the surrounding background. The laboratory tests also indicated that the use of contrast variation, as opposed to changes in image brightness, for determining spray density produced more repeatable results under a wider range of lighting conditions. The use of a contrast-based system therefore allows for better correlation of results from different test sessions than is possible with a brightness-based system. This has the advantage that testing can be conducted at different times, under different lighting conditions, while maintaining confidence in the repeatability of results.

In order to measure the change in contrast due to spray, chequered target boards were used. These target boards consisted of a grid of alternating black-and-white squares. An image of each board was taken immediately prior to the boards being obscured by spray and the contrast of each grid square in this 'control' image defined as 100% contrast. A series of images was then recorded with the target boards partially obscured with spray and the contrast of each grid square again calculated. The change in contrast of each grid square was then calculated as a percentage of the contrast of the control image, thus giving a relative value which can be compared to other results taken under different conditions.

2.1 Hardware

The main hardware components of the spray-measuring system were a pair of CCD cameras, associated video capture cards and the two chequered target boards. The CCD cameras had manually adjustable iris and shutter speed so that they did not automatically correct for the increased image brightness when spray is present. The video capture cards were installed in PCs and set to capture video at a rate of 12.5 frames-per-second at a resolution of 192x144 pixels. The images were 8-bit (256 level) greyscale. Each video sequence recorded contained 100 frames (8 seconds of video), of which approximately half were used for analysis in determining the effectiveness of the spray-reducing devices.

The target boards measured 3.2m horizontally and 2.4m vertically. Each board had been divided into a black-and-white chequered grid of 195mm squares (192 squares on each board). Vertical borders of 40mm and horizontal borders of 30mm surrounded the chequered region to prevent light diffusing around the target boards into the recorded region.

2.2 Software

Video capture was performed with software supplied with the video capture boards. After a video sequence was captured, it was converted into a series of bi-mapped images. As with the captured video, these images were 8-bit greyscale.

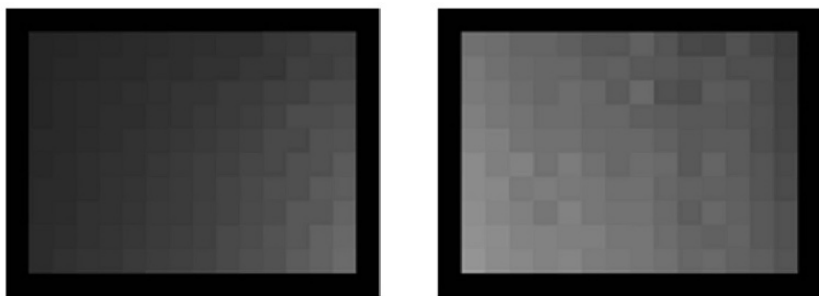


Fig. 1 Examples of time-averaged contrast change images (left and right-hand sides of a passing truck)

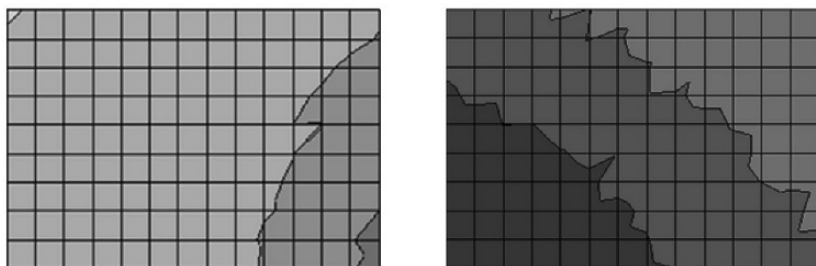


Fig. 2 Examples of spray-density contour plots (left and right-hand sides, corresponding to the images in Fig. 1)

Custom software was written for the analysis of the video images. The analysis relied on the change in image contrast due to spray obscuring the target boards. The first image of each sequence was taken as the control image and the analysis software assigned each grid square in the control image a contrast value of 100%, after calculating an absolute value of contrast. Here contrast is defined as the ratio of the brightness of the centre square to the average brightness of the squares above, below, left and right of the centre square.

Subsequent images were compared relative to the control image and the percentage difference in contrast of each grid square, relative to the corresponding square of the control image, was taken as the change in contrast for that square. The average change in contrast for each square over the series of images was also calculated to form an image of time-averaged contrast change (an example of the time-average contrast change, for the left and right-hand sides of the vehicle, is shown in Fig. 1, where the white regions indicate areas of decreased contrast). From the averaged image, the overall change in contrast of the image was calculated and topographical plots were formed to show the density distribution of spray. An example spray-density contour plot is shown in Fig. 2. As images were recorded from each side of the test vehicle simultaneously, an average of the left and right-hand sides of the vehicle was also calculated. The average of the two sides was used for comparisons between different test runs and vehicle configurations as the effects of varying wind conditions was shown to be greatly reduced.

Results were processed immediately after each run to provide rapid feedback as to the effectiveness of a spray-reducing device.

After the initial analysis of results, it was observed that the results were being influenced by the 'shake' of the cameras as the test vehicle passed. As the location of the grid squares were moving relative to the boundaries of the images, this generated apparent changes in contrast. To overcome this problem, two routines were added to the software. The first routine located the boundaries of each individual square from the control image while the second routine 'tracked' the grid squares as they moved with camera shake - a form of digital image-stabilization. With the implementation of these two routines, it was found that the results were considerably more repeatable.

3. Testing Of Spray-Reducing Devices

3.1 Test Track

Testing was conducted at the Anglesea Complex Proving Ground, in Victoria, Australia. A 3.2km oval track at the Complex was employed for the tests as it incorporates a 200m section of wetted track and allowed the test vehicle to maintain a steady 100km/h through the test section. In the test section, the track surface is fine-grade asphalt and there is a downhill gradient of 1-in-200 in the direction of travel and a cross-slope of 1-in-75 from right-to-left. Surface wetting was obtained from a series of sprinkler heads spaced at 5m intervals along the high (right) side of the track. The sprinkler flow rates were independently variable and were set to provide the maximum flow rate without impinging on the test vehicle as it passed. All of the sprinkler heads rotated through 120° to provide a reasonably even coverage of water. The sprinklers were operated for 30 minutes before testing to allow the water distribution to stabilize and then run continuously for the remainder of testing. Due to slight undulations of the track surface, the water depth varied from <1mm to 3mm in the test section. A short duration after each run ensured the track was repeatably saturated.

3.2 Test Procedure

An overview of test layout and procedure can be seen in Figs. 3 and 4. The cameras were mounted on tripods at a height of 1m (similar to the eye level of a

driver seated in a conventional car) and laterally aligned with the centre of the corresponding target board. The target boards were mounted with their lower edges 800mm above the track surface with a distance of 3.5m between the two boards, allowing a clearance of 500mm between the extremities of the 2.5m wide test vehicle as it passed between them. The target boards were placed at the end of the wetted section of track with the cameras placed 75m from the target boards. This left approximately 125m of wetted track for the complete wetting of the underside of the test vehicle before it passed the cameras.

Recording of the video images was triggered by an infra-red relay switch 20m before the truck passed the cameras.

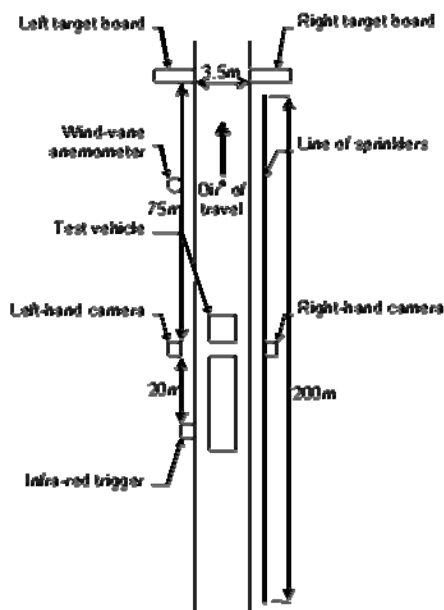


Fig. 3 Test overview

3.3 Test Vehicle

All tests were conducted with a Freightliner FL106, conventional-cab, tandem-axle tractor unit and a Lucar 4.0m, tandem-axle box trailer. The test vehicle is shown in Fig. 4.



Fig. 4 Test vehicle and track

3.4 Spray-Reducing Devices

Three configurations were tested. ‘Configuration A’ was the baseline configuration of the truck, as supplied for testing. This configuration included mudguards fitted to the drive wheels and untextured rubber mudflaps fitted behind all wheels. ‘Configuration B’ was the same as the baseline configuration but with the fitment of commercially-available textured mudflaps to the drive and trailer wheels (necessitating the removal of the untextured flaps). ‘Configuration C’ was formed by the removal of all flaps and the drive-wheel mud-guards.

3.5 Test Method

Several recorded runs of each configuration were performed to determine the repeatability of the results. The vehicle speed was held at a constant (and speed governed) 100km/h through the test section. Video capture was triggered 20m before the truck passed the cameras and continued for 8 seconds, at the end of which the truck was 125m past the target boards. This allowed the first image of each video sequence to be used as the control image and the build-up and dissipation of spray to be recorded. At regular intervals throughout the day, ambient temperature and humidity readings were taken. Wind speed and wind direction were also measured for each run.

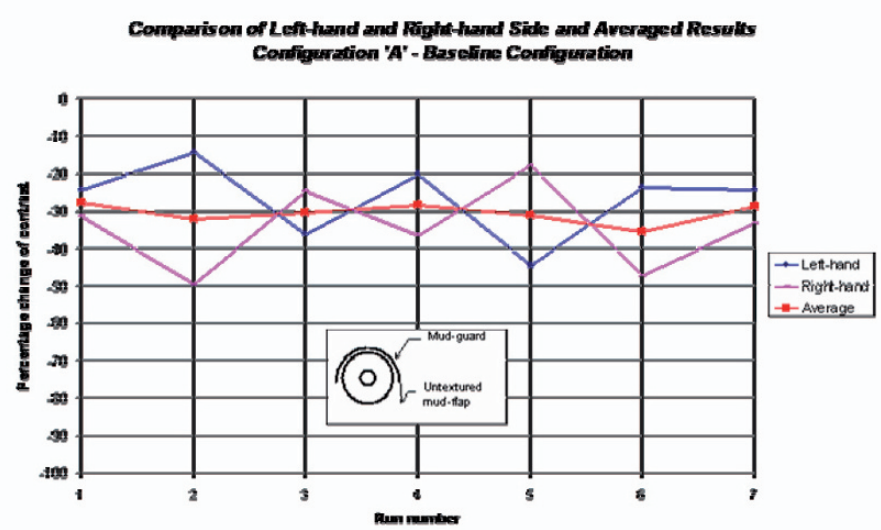


Fig. 5 Results of Configuration 'A'

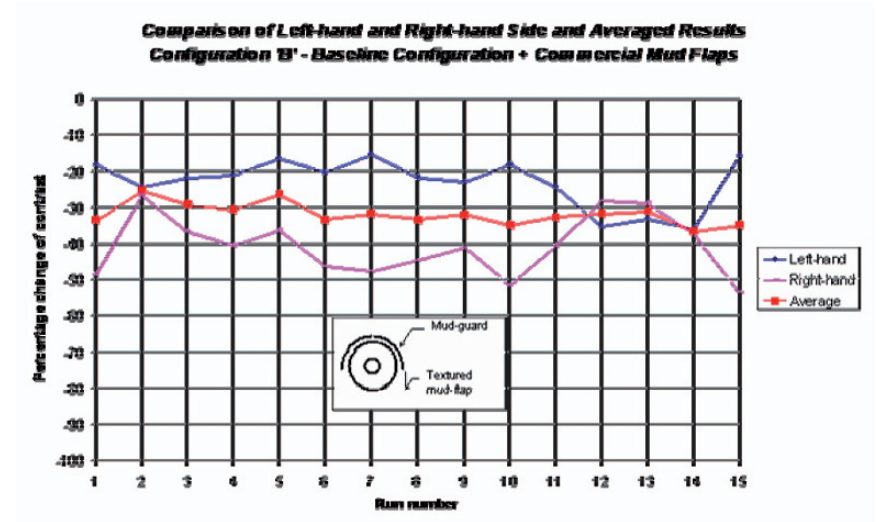


Fig. 6 Results of Configuration 'B'

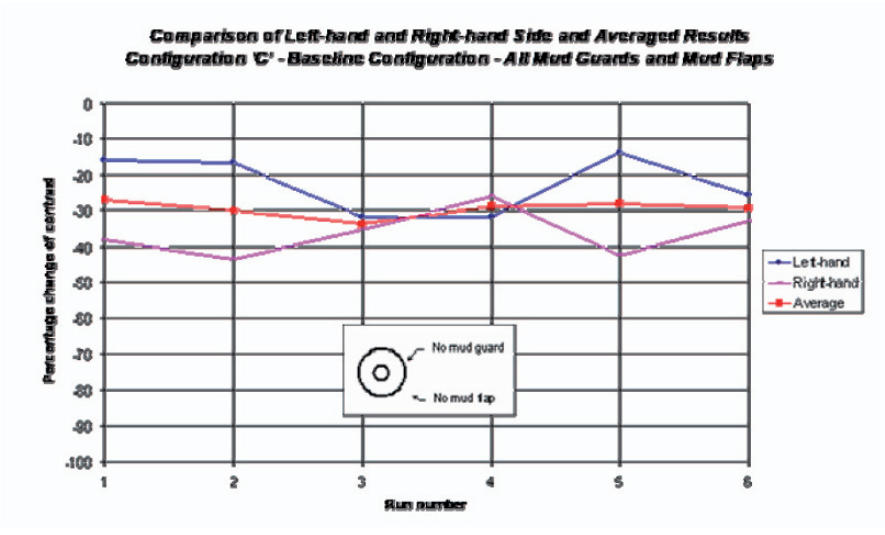


Fig. 7 Results of Configuration 'C'

3.6 Results and Discussion

The repeatability of results with varying ambient conditions was found to be very good. It was also found that the averaging of the left and right-hand side results proved very effective in reducing the effects of varying wind direction and wind speed, although it should be noted that winds were light (<10km/h) and wind direction was relatively constant. No significant changes in ambient temperature or humidity were recorded.

The graphs of Fig. 5 to 7 show the results for each of the three configurations tested. Each graph shows the percentage change in contrast due to spray for the left and right-hand sides of the truck and the average change in contrast of both sides for a number of runs. The graphs show that while the results from the left and right sides varying considerably, the averaged results remain relatively consistent. This is particularly evident with Configuration A - the percentage of spray recorded on the left and right hand sides varied considerably but the average of the two sides remained almost constant at a change in contrast of -30%.

This is further evidenced from Table 1, which gives a summary of the results. Values given are the calculated mean percentage change in contrast, the population standard deviation from the mean, 95% confidence intervals from the mean and the 95% confidence intervals as a percentage of the mean value. These values are given for the left-hand side of the truck, the right-hand side of the truck and the average of the left and right hand sides of the truck.

From the 95% confidence intervals it can be seen that it is advantageous to measure the spray from both sides of the truck simultaneously and average the

results. Referring to Configuration A, the 95% confidence intervals for the left and right hand sides are 7.1 and 7.9 (or 26.3% and 22.9% of the means) respectively. However, the 95% confidence interval for the averaged result is 1.9, or 6.1% of the mean value. The results for the other configurations tested also show similar patterns.

The results also indicated that there was very little variation in the quantity of measured spray between each configuration. As the differences in results are all within the 95% confidence intervals, no conclusions can be made as to which configuration performed best. These results were also confirmed by spectators during testing and by subsequent viewing of video tape recorded during the testing.

Figure 7 shows a plot of the typical variation of spray density against the test vehicle distance from the camera. At a distance of 0m, the front of the truck is level with the camera, at 75m the front of the truck is level with the target boards and at 200m the truck is 125m past the target boards. If looked at from right-to-left, the graph is an indication of the variation in visibility that a motorist can expect to see when approaching a truck and then overtaking that truck. Initially, there is a steady increase in the spray density, the rate of which increases as the back of the truck is approached. After passing the trailer wheels the spray density begins to decrease but there is a slight increase in density as the drive wheels are passed. Beyond this point there is a rapid decrease in spray density until the truck is completely passed. (The reduction in contrast near a distance of 80m is believed to be due to the target boards and the influence they have on the flow field and the truck passes by them.).

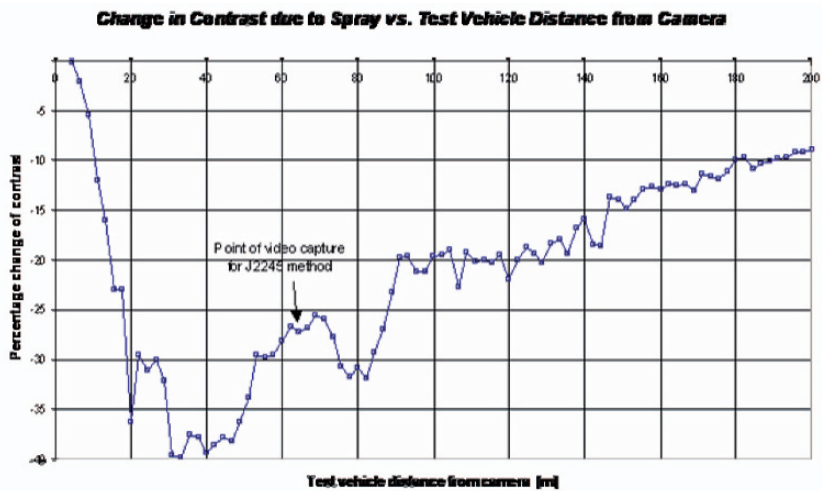


Fig. 8 Typical variation of change in contrast due to spray with test vehicle distance from camera

Table 1 Summary of results

Configuration 'A'			
	Left	Right	Ave
Mean %ADC	-26.8	-31.4	-30.6
StdDev	9.5	10.6	2.5
95% Confidence	7.1	7.9	1.9
(95% CtlMean)*100	26.3	22.9	6.1
Configuration 'B'			
	Left	Right	Ave
Mean %ADC	-23.0	-40.6	-31.8
StdDev	6.5	8.1	2.9
95% Confidence	3.3	4.1	1.5
(95% CtlMean)*100	14.3	10.1	4.6
Configuration 'C'			
	Left	Right	Ave
Mean %ADC	-22.5	-36.3	-29.4
StdDev	7.5	5.9	2.1
95% Confidence	6.0	4.7	1.7
(95% CtlMean)*100	26.7	13.1	5.7

3.7 Comparisons of Technique with Recommended Practices

A Society of Automotive Engineers (SAE) standard exists for spray measurements – J2245 [8]. There is one main difference between the spray-measuring system described here and [8] - the J2245 method records four frames of video at 30 frames-per-second when the front of the truck is 10.67m (35ft) from the target boards, whereas the method described here records 100 frames of video at 12.5 frames-per-second. The J2245 method therefore provides an average over a very short period of time (4/30ths of a second). This has one main advantage over a longer recording period. The short time period means that the results are relatively instantaneous and are therefore not greatly effected by the camera ‘seeing’ different regions of the spray cloud as the camera-to-truck distances changes over the time of recording (although this does simulate what a passing motorist will see). However, as Fig. 8 shows, the density of spray changes rapidly in the region where the four frames are recorded (10.67m before the target boards or about the 65m mark). Therefore, small variations in the point of recording can produce significant variations in results. Also, the spray clouds being measured are highly turbulent and time-variant. From a study of video recorded during testing, it is believed that the period of oscillation of the spray clouds can be as long as one second. This indicates that several seconds of video needs to be analyzed for good statistical significance of results. Doing so should also reduce the number of runs required to attain the same degree of repeatability.

There are other minor differences between the two methods (such as the distance from cameras to target boards, the calculation of grid square contrast values and vehicle speed in the test section) but none of these are expected to have a significant influence on the differences in results.

4. Flow Field Measurement – An Overview

As supporting documentation to the spray work, flowfield mapping was undertaken utilizing dynamically calibrated “Cobra” probes which could provide information on the time-average and transient flow fields around commercial vehicles. Details of the probe systems can be found in [9] and they were found to be suited to use during on-road and wind-tunnel testing, particularly the ability to measure the local pressure in a turbulent flow field away from a solid surface. In the outdoor environment, the much improved durability and long-term stability compared to alternative methods (such as hot-wires or laser-Doppler systems) was considered very beneficial. The on-road tests were performed by driving the truck (with various modifications, including full side skirting almost down to the road surface) past banks of probes. The tunnel tests involved 3-component traverses around and under a one-tenth model of the truck. For each measurement point, the following data were recorded: u , v and w components of velocity; static pressure; reference dynamic pressure; temperature; and barometric pressure. As the time history of each variable was recorded, the turbulence intensities and Reynolds stresses were also calculated at each point. Details of the test methods and results for track and wind-tunnel environments can be found in [10]. A selected result that illustrates the influence of the flow changes associated with aerodynamic modifications is shown in Fig 9. Here the velocity vectors relative to the moving vehicle are plotted (i.e. the magnitude of the freestream velocity has been removed from each point to allow for comparison with the on-road pass-by measurements) and are nondimensionalised by the freestream velocity. These measurements were for a horizontal plane that was equivalent to 250mm (in full scale) above the road surface.

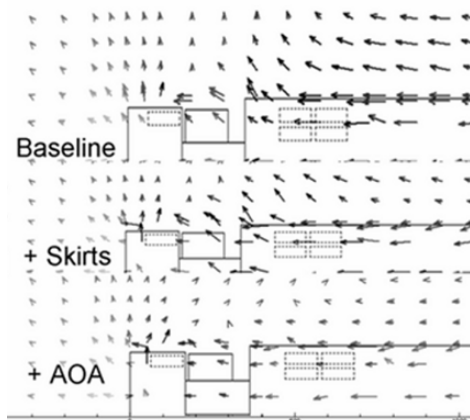


Fig. 9 Velocity vectors, resolved in the horizontal plane and relative to a moving vehicle, at equivalent road elevation of 250mm, adapted from [10]. Data taken from a one-tenth scale model of the full-size truck.

5. Discussion

The rapidly changing velocity field alongside trucks is apparent and highlights the need for instrumentation with a high frequency response when utilizing the pass-by method of testing. From the track tests [10] the addition of skirts was noted to reduce the local pressure alongside the truck and to also reduce the velocity in the same area. This reduction and modification in the velocity is seen in the tunnel results above and most noticeable is the reduction in lateral air velocity components in the area between the rear of the cab and the front of the trailer face when a cabin mounted add-on aerodynamic device (AOA) was fitted. This flow modification when (correctly matched) AOAs are used on trucks hauling relatively high loads is considered to offer significant benefits. From the track work it was seen to prevent the spray from being ‘carried along’ with the truck. From a combination of the tunnel and track work, the reduction in lateral flow emanating from the drive wheels could clearly be seen to reduce the spray in this region.

Although not part of the spray testing program it is instructive to consider Fig. 10. This aerial photograph shows a “road train” on a dry dirt road in the Northern Territory of Australia. Whilst the vehicle has no aerodynamic device and a quite different cabin to the vehicle in the spray tests, the visualization by the dust clouds offers insight into the flow field.



Fig. 10 Dust flow visualization from a cattle road train

Several features are evident. The flow to either side of the vehicle is not symmetrical – in either a transient or time-averaged manner. It appears that there is a

slight crosswind component of the atmospheric wind causing a significantly larger wake on the leeward side of the vehicle. This emphasizes the need for averaging even on days of relatively low winds. Secondly some of the flow below the stagnation point on the trailer front face is deflected down and into the gap between the cabin rear and the trailer, where some flow emerges laterally in the vicinity of the drive wheels – this is evidenced as the dust-free flow in this area. Prior work has established that the drive wheels are a major source of spray and the lateral flow at this point will cause the spray from the drive wheels to be laterally deflected across the areas to the left and right of the truck and into the line of sight of the passing motorist.

5. Conclusions and Recommendations

The spray-measuring system has been shown to be able to repeatably measure truck-induced spray. Although there were no major variations in ambient conditions during testing, the results indicate that the system effectively averages results to reduce the effects of varying ambient conditions, particularly cross-winds.

The results obtained from track testing include a measure of the overall spray cloud density, a picture of the distribution of spray within the viewable area and an indication of the variation of spray density as would be seen by a passing motorist. The spray-measuring system could also be used to measure the extent of other particle clouds, such as diesel or industrial exhaust smoke, with minor modification.

The road and tunnel flow field measurements enabled new insights into the low-level flow field which is considered to be an important “driver” for the spray generated by trucks on wet roads. In particular it was noted that the lateral flow was considerable when there was no AOA cabin-mounted aerodynamic device for a truck hauling a high load (but less so when trailer skirts were fitted). The work demonstrated the correct use of cabin devices can reduce the displacement of spray laterally and into the potential path of any passing vehicle. It has also been shown on several occasions that both the skirts and cabin-mounted devices reduce drag of trucks via wind-tunnel and road tests [e.g. 12]. Thus the use of correctly matched cabin-mounted devices will not only reduce drag; they will also reduce the lateral flow that drives spray from wet roads into the areas of concern for passing motorists. This offers the trucking community an added bonus that will result in enhanced opinions from the passing motorists.

6. Acknowledgments

The authors would like to thank the following companies and individuals, without whom this program could not have been undertaken: Mercedes-Benz Australia, Australia Post, Drs. Garry Zimmer and Juliette Milbank and Mr Tim Paton. The financial support and encouragement of Mr. Keith Seyer of the Federal Office of Road Safety and Mr. Brian Wells of the Australian Automobile Association is gratefully acknowledged.

7. References

1. C Baughan, B Hedges and J Field, "A National Survey of Lorry Nuisance", TRRL Report SR 774, Transport and Road Research Laboratory, 1983.
2. Anon, "Update on Status of Spray Suppression Technology for Large Trucks", Report US to Congress, March 2000. Available from: <http://www-nrd.nhtsa.dot.gov/PDF/nrd-12/SplashSpray.pdf>
3. Dumas and Lemay, "Splash and Spray Measurements and Control: Recent Progress in Quebec", in Proc. The Aerodynamics of Heavy Vehicles; Trucks, Buses and Trains, Springer 2004, ISBN-10: 3540220887.
4. C Baughan, "Water Spray from Road Vehicles: Measuring the Effectiveness of Spray Reductions," Proc. Vision in Vehicles - III, A.G. Gale et al, ed., Elsevier Science Publishers B.V., pp. 275-282, 1991.
5. J Scheltens and A Luyombya, "Spray Cloud Measurement System Based on Computer Analysis of Video Images", SAE Technical Paper 872277, 1987
6. C Gorte, R Joyner, K Pedersen and C McConnell, "The MVMA Investigation Into the Complexities of Heavy Truck Splash and Spray Problem", Proc. 10th International Technical Conference on Experimental Safety Vehicles, U.S. Dept. of Commerce National Technical Information Service, pp. 848-863, 1985.
7. Puclin A, "Quantitative Truck Spray Measurements", Masters by Research Thesis, RMIT University, Melbourne, Australia, 1997.
8. Anon, "Recommended Practice for Splash and Spray Evaluation - SAE J2245", SAE Handbook 1995, pp. 28.01-28.08, 1995.
9. Watkins S, Mousley P and Hooper J, "Measurement of Fluctuating Flows Using Multi-Hole Probes", Proc. 9th Int. Cong. on Sound and Vibration, University of Southern Florida and NASA, July 2003. Details of the instrumentation can also be found at: <http://www.turbulentflow.com.au>

10. Mousley P and Watkins S, "A Method of Flow Measurement about Full-Scale and Model-Scale Vehicles", SAE Technical Paper 2000-01-0871 also in SAE Transactions – Journal of Passenger Cars, 2000, ISBN 0-7680-0574-4.
11. Saunders J, Watkins S, Hoffmann P and Buckley F, "Tractor-Trailer Aerodynamic Devices; Fuel Saving Predictions and Comparison of On-Road and Wind Tunnel Tests", SAE Technical Paper 850286, from International Congress of the Society of Automotive Engineers America, Detroit, February 1985.

Experiments on Spray from a Rolling Tire

Charles Radovich and Dennis Plocher

Department of Aerospace & Mechanical Engineering
University of Southern California, Los Angeles, CA 90089
radovich@alumni.usc.edu, plocher@usc.edu

Abstract A novel laboratory apparatus has been built to understand the key mechanisms behind spray emerging from a rolling tire. Water leaving the tire from a single circumferential groove is analyzed using high-speed imaging. Visualizations reveal the formation of thin sheets of water connecting the roadway and the tire that eventually break into a droplet field. It is proposed that sheet breakup is the result of hydrodynamic instability. There is a preferred wavelength for disturbances on the sheet. After breakup, this preferred wavelength is preserved in slender ribs (or ligaments) that continue downstream until they disintegrate into droplets. Weber numbers based on the water density, tire circumferential speed and groove width vary in the range 1,900 – 60,300. The transition from sheet to spray is accelerated with increasing Weber number. Farther downstream, it is shown that droplet size and velocity distributions can be quantified as a function of Weber number.

1 Introduction

Tires moving through standing water create splash and spray which can decrease visibility and prove hazardous for other vehicles. Spray is defined here as water that has been forced through the tread pattern in the tire patch (contact patch between the tire and the pavement), thus, “processed” by the tire.

Our visualizations show that tires produce water spray as a result of the formation and breakup of thin sheets of fluid. An example spray image is shown in Fig. 1.1, taken using a laser sheet for illumination (~10 nanoseconds exposure time). The tire on top (in the image) has a single rectangular circumferential groove 11 mm wide by 5.6 mm deep. The bottom tire has no groove. The peripheral speed of the tires is about 12 m/s, and the volume flow of water delivered upstream of the tire contact patch is just sufficient to fill the circumferential groove.

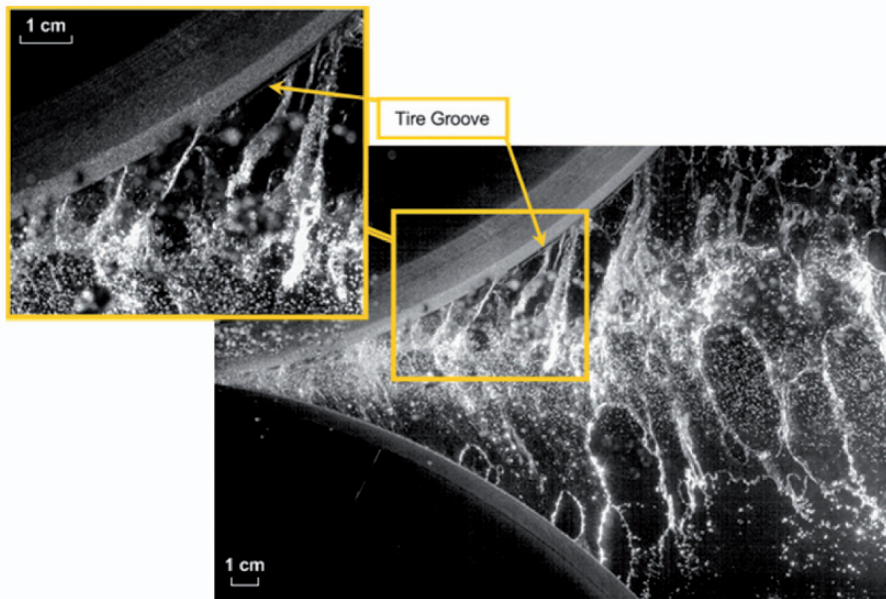


Fig. 1.1 Spray image using laser sheet illumination (~ 10 ns pulse time)

The laser illuminated image contains bright, irregular regions containing water that has left the circumferential groove. These irregular regions appear attached to the tires by ligaments and by thin sheets of water occupying the regions between the ligaments. There are breaks in the sheets and the smallest droplets appear to form along these rupture boundaries. Referring to the water leaving the circumferential groove as the central “jet,” the droplet formation process appears to be the combination of the instability and breakup of this central jet and the breakup or tearing of the thin sheets connecting the central jet to the tires.

Downstream from the contact patch, the spray ultimately disintegrates into a droplet field. Information on the distribution of sizes and velocities of these droplets is desired. This paper details the steps taken towards imaging and analyzing the spray immediately leaving the contact patch and the droplet field farther downstream. In brief, droplet size estimates come from counting the number of pixels a droplet occupies in an image. After determining the size distribution, images are “filtered” for specific size ranges and digital particle image velocimetry (DPIV) is used to estimate droplet velocities as a function of droplet diameter.

2 Description of the Experiment

2.1 Justification for the two-tire geometry in the laboratory

In order to correctly model a tire rolling over a wet road, the physics of the tire patch must be understood. There are two reference frames applicable, as shown in Fig. 2.1. In the first reference frame, an observer watches a tire roll along the wet pavement at a velocity U_{truck} . In the second reference frame, the observer rides with the rotating tire and sees water coming at the tire with velocity U_{truck} . Now, using the principle of symmetry, a second tire represents the road, as shown in

Fig. 2.2. This principle of symmetry and a reference frame riding with the car is used to model the tire/water interaction in the laboratory.

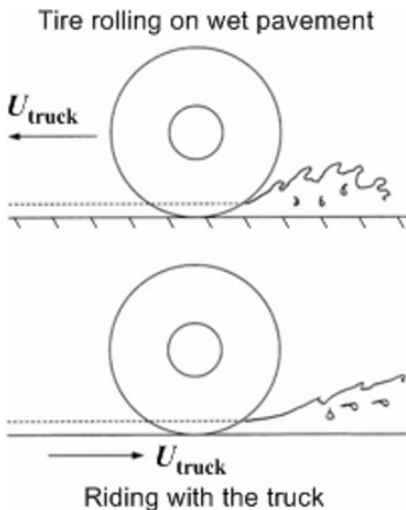


Fig. 2.1 Two reference frames for a tire rolling over wet pavement

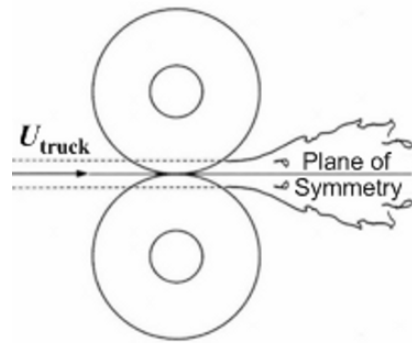


Fig. 2.2 Using principle of symmetry, a second tire represents the pavement

2.2 The Tire Spray Simulator

The apparatus for studying tire spray is shown in Fig. 2.3. It consists of two rotating tires pressed together to form a tire patch, a system to deliver water at the peripheral speed of the tires and a means for visualizing spray coming from the tire patch.

Two wheels are suspended by swinging arms with tires touching. The tires (P-175/70 R13) are pressed together to form the contact patch and are held in place by a spring-shock that is positioned using a stepper-motor-controlled lead screw. A Sensotech load cell measures the force between the tires. The left-most wheel in Fig. 2.3 rotates by means of a chain drive and a Leeson DC electric motor and motor-controller. The tire to the right is driven by contact at the tire patch.

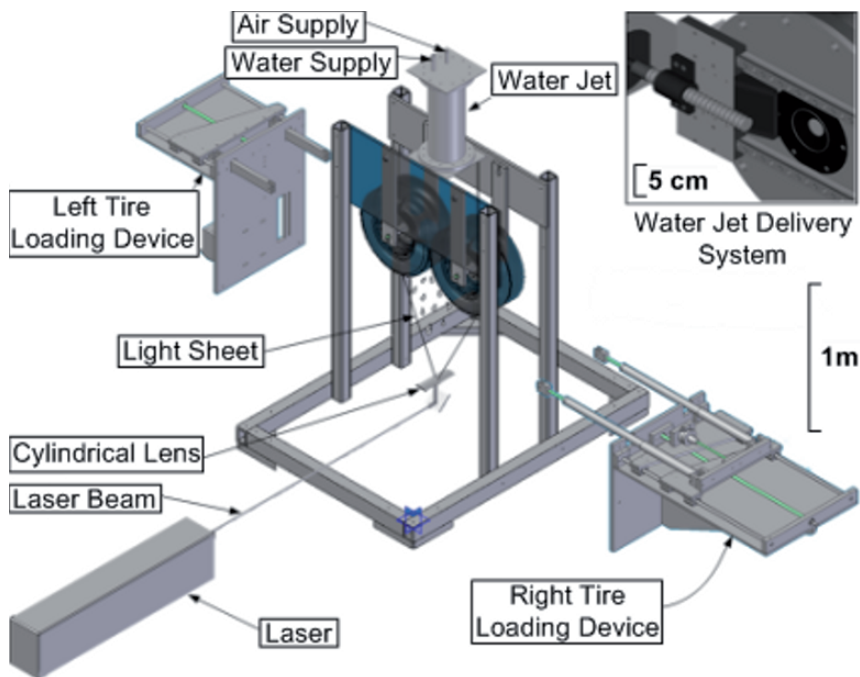


Fig. 2.3 The tire spray simulator

Wheel speed can be set in the range 3 m/s – 25 m/s. Precise wheel speed is measured by counting time intervals between pulses produced by a (stationary) Hall-effect pick-up and a toothed ring mounted on each wheel hub. The measured circumference of the grooved (*g*) and non-grooved (*s*) tires was $C_g = 1778 \pm 5$ mm and $C_s = 1810 \pm 5$ mm. Since the uncertainty in the time interval is on the order of microseconds, error in the estimate of tire speed, $\Delta U/U$, comes from uncertainty in knowledge of the tire circumference and is of the order 3×10^{-3} or less.

Visualizations of the tire spray are captured using a video camera set to record at 1600 fps and 5 μ s exposure time. Recorded images are 400 by 1260 pixels in resolution. The spray field can be illuminated using either a laser sheet or backlighting. Laser sheet illumination allows for sharp, detailed images; however, pixel intensity information is often lost due to camera saturation as a result of specular reflection of light from the water surface. A more suitable means for imaging, and the method used for the remaining results in this paper, is backlighting.

Figure 2.4 displays a side view of the experimental setup showing the orientation of the camera, tires, frosted (diffusive) glass and light source.

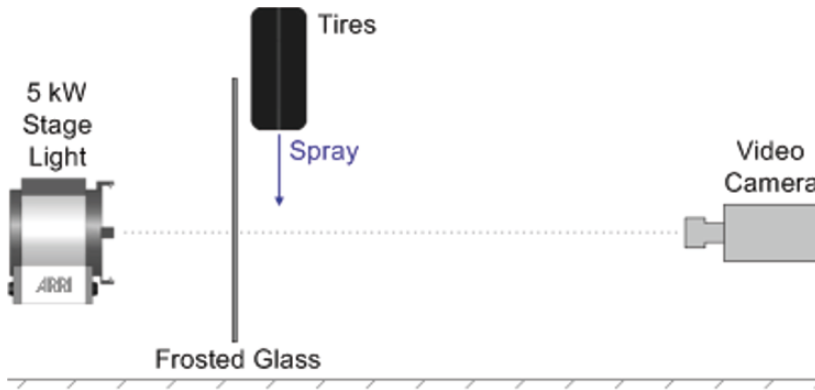


Fig. 2.4 Experimental setup for backlight imaging

2.3 Droplet Size Calibration

A series of calibration images were taken to verify that droplets of various sizes could be identified at all locations in the image field. A syringe pump, fit with a 21G hypodermic needle, delivered a single water droplet of a known size to the image field. A variation in droplet sizes was simulated by moving the camera farther from (or closer to) the free falling droplet. For each image of a droplet, a background image was recorded just before the droplet entered the field of view. This background image is used to account for and to correct any non-uniformities in the backlighting. Figure 2.5 shows an image of the prescribed droplet paired with its background image.

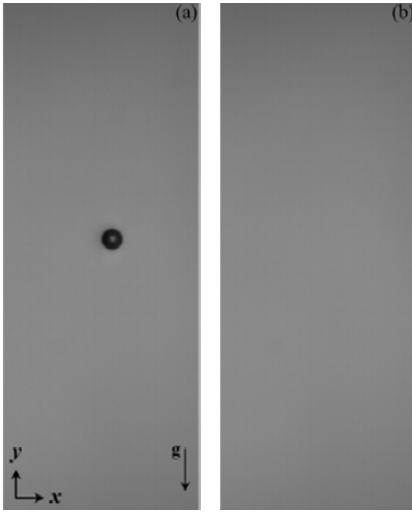


Fig. 2.5 (a) Image of a freefalling water droplet ($D = 2.84$ mm), (b) Background image, taken just before the droplet entered the field of view

Figure 2.6a shows a surface map of the image intensities recorded for a single droplet. Intensity values are expressed as a fraction of the average background intensity. The curvature of the surface is evidence of a non-uniformly lit background. In order to correct for this gradient, a linear gain correction matrix, $G(i, j)$, based on the background image was calculated and applied to the original intensity map $I'(i, j)$. Indices i and j refer to each pixel in the image field. Figure 2.6b shows the gain corrected intensity map $I_c(i, j)$. The final image used for analysis, $I(i, j)$, is obtained by subtracting the average background intensity from I_c and performing an inversion of the intensity map; Figure 2.6c. Equation Set 1 outlines the gain correction procedure.

$$\begin{aligned}
 G(i, j) &= \overline{BG} / BG(i, j) \\
 I_c(i, j) &= G(i, j) * I'(i, j) \\
 I(i, j) &= -1 * (I_c(i, j) - \overline{BG})
 \end{aligned}
 \tag{1}$$

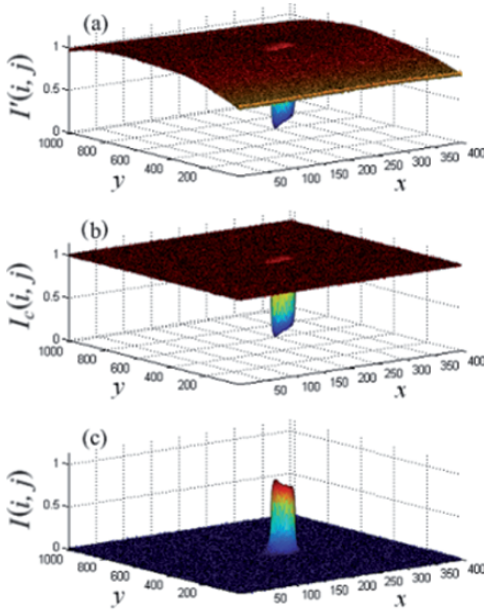


Fig. 2.6 Image intensity maps for (a) the raw image, (b) gain corrected image, (c) gain corrected and inverted image used for analysis



Fig. 2.7 Gain corrected and inverted image of a water droplet

An example of a gain corrected and inverted image is shown in Fig. 2.7. This figure has a single droplet near the center of the camera field of view. Back-lighting transmitted through the droplet contributes to the dark spot in the center of the inverted droplet image (seen as a bright spot in Fig. 2.5a). Droplets can pass through any part of the image during a tire spray experiment. In order to verify that droplets are identifiable at all locations in the image, the calibration procedure was repeated for droplets located at the top and bottom of the image field.

The ability to resolve droplets of various sizes was also verified. The presence of smaller (larger) droplets was simulated by using the same 21G syringe pump and moving the camera farther away from (closer to) the apparatus. At each camera location, the prescribed droplet occupied a certain number of pixels. Observed droplet sizes ranged from 55 to 1380 pixels in area.

Figure 2.8 shows the intensity surface maps for four different droplet sizes at the top, center and bottom of the image field. This figure is a composite surface map with each droplet coming from a separate test. The presence of a droplet creates an intensity “mountain” with a broad base and narrow peak. The number of pixels retained in this mountain leads to the droplet size estimate. The black plane represents a cutoff threshold that determines which pixels to keep or to discard. Pixels with an intensity value at or above the threshold will be retained. Increasing or decreasing the threshold value will change the number of preserved pixels and subsequently alter the droplet size estimate.

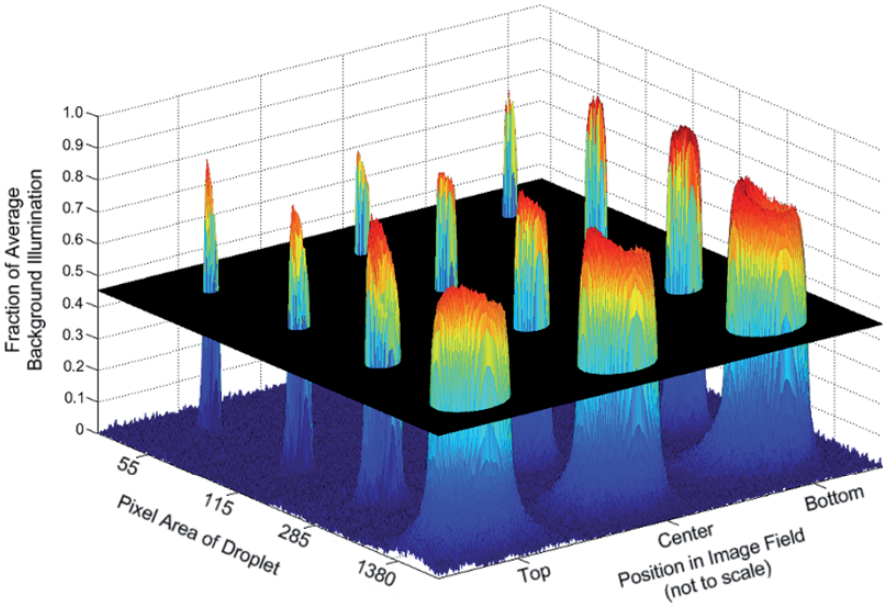


Fig. 2.8 Image intensity maps of four droplet sizes at the top, center and bottom of an image

Figure 2.9 illustrates the steps taken to estimate droplet size by counting the number of pixels a droplet occupies. Gray-scale images are reduced to black and white (binary) images after applying a threshold; Figure 2.9a. Pixels that are preserved are turned white (1) and everything else is set to black (0). Large droplets often allow a partial amount of light to transmit through, resulting in a “hole” at the center of the droplet. If a hole exists, it must be filled before counting pixels and determining the droplet size; Figure 2.9b. Once all droplets have been filled, the remaining pixels can be counted; Figure 2.9c.

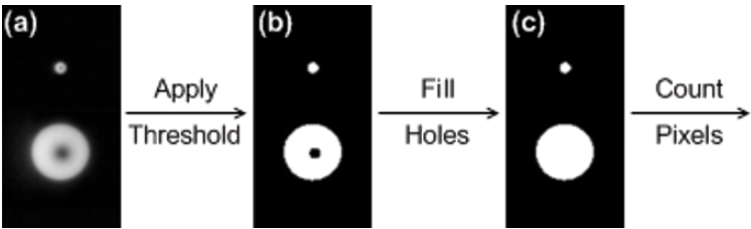


Fig. 2.9 Image processing used to determine the size of a droplet

Figure 2.10 shows the effect on the droplet diameter estimate when various threshold values are used. The abscissa represents the droplet area in terms of pixels; the ordinate is the estimated diameter calculated using $D = \sqrt{(4A/\pi)}$ and the

appropriate mm/pixel factor for each camera location tested. The size of the droplet is 2.84 ± 0.09 mm in diameter, based on the average weight measured for a series of droplets. The experimental results provided in this figure clarify what threshold value is needed to estimate a droplet diameter within 5%, and to what extent various droplet sizes can be resolved with a single exposure. There is an increased sensitivity to minor changes in the threshold value for smaller droplets since these images have fewer pixels compared to the larger droplets. For all droplet sizes tested, an accurate estimate of the diameter was achieved using a threshold value equal to 45% of the average background illumination.

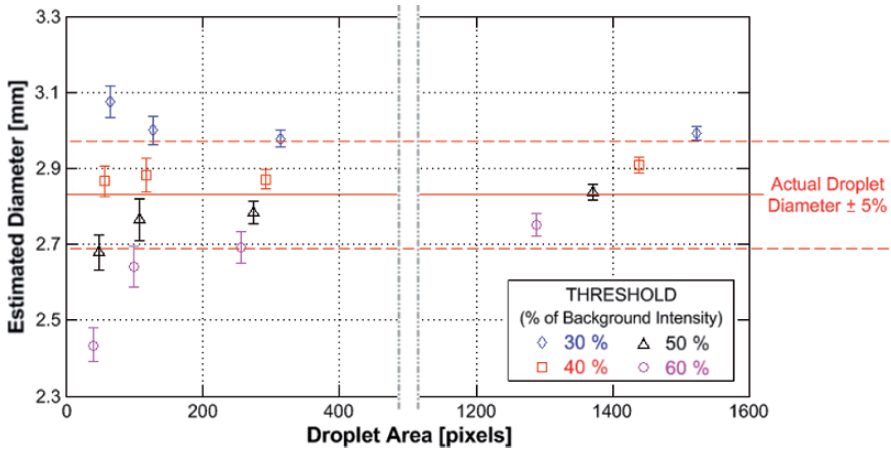


Fig. 2.10 Droplet diameter estimates for several droplet sizes and various threshold values

It was also observed that, in the spray plane, droplet images have a high contrast ratio compared to images outside of the focal plane. During the initial water delivery, spray may fall outside of the contact patch or create water spots on the frosted glass plate. These artifacts are undesirable for analysis as they are outside of the spray plane (the region downstream of the contact patch where the camera is focused). Therefore, applying an intensity cutoff threshold to the image retains pixels within and discards artifacts away from the region of interest. Previous works by Marmottant and Villermaux [1] and by Eastwood *et al* [2] illustrate a similar effort towards droplet detection and sizing.

3 Results

3.1 Sheet Formation and Break-Up

The Weber number is the most important parameter governing the formation of the water sheet and subsequent decay into a droplet field. This quantity is a ratio of the distorting force associated with fluid inertia to the (largely) restoring effect of surface tension (Equation 2). The Weber number is defined using the width of the tire groove ($h = 11$ mm), the density of water ($\rho = 998$ kg/m³), the speed of the water jet (U) and surface tension for water against air ($\sigma = 0.073$ N/m).

$$We = \frac{\rho h U^2}{\sigma} \quad (2)$$

A backlit image of the spray field produced at $U = 6$ m/s ($We = 5,400$) is shown in Fig. 3.1; the direction of flow is from left to right. The upper tire has a single circumferential groove; the lower tire has a buffed smooth surface. The field of view in this image is approximately 26.1 by 8.3 cm with the end of the contact patch visible on the left edge. Fluid leaving the contact patch remains attached to both tires and forms a thin sheet as the tires separate. The thickness of the sheet was not measured directly, however, it is estimated to be on the order of <1 mm. There are waves on the sheet that appear as dark rib structures. These ribs exhibit a periodic nature, suggesting a natural instability in the fluid flow. The sheet ultimately breaks up into a droplet field as it moves away from the contact patch.

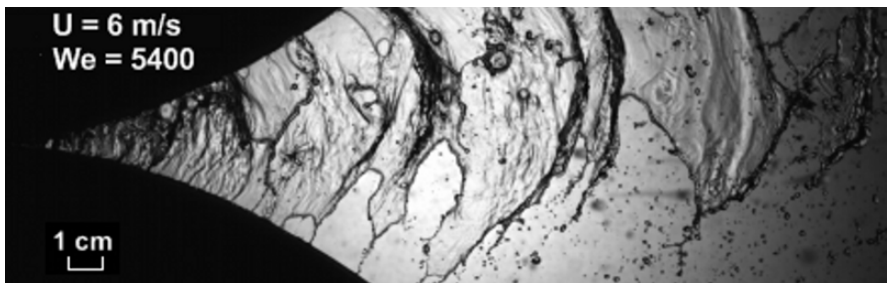


Fig. 3.1 Tire spray pattern at $U = 6$ m/s, $We = 5,400$

The development of the liquid sheet is sketched in Fig. 3.2, as it would appear at several cross planes looking upstream. The water passes through the tire patch

inside the groove. At the downstream end of the tire patch the treads begin to move apart following the arc of the rotating tires, with the water exiting the tire patch as a jet attached to the groove on one side and to the smooth tire on the other side. The groove begins to drain and air flows in to fill the developing void. Sheets connect the jet to the smooth tire and to the water remaining in the groove. The jet takes on a sinuous shape (instability) as can be seen at the extreme left in Fig. 3.1. Farther downstream, the sheet begins to separate from the surfaces of the tires. As breaks appear in the sheet, the sinuous jet forms into slender ribs (ligaments) orientated cross-stream to the flow.

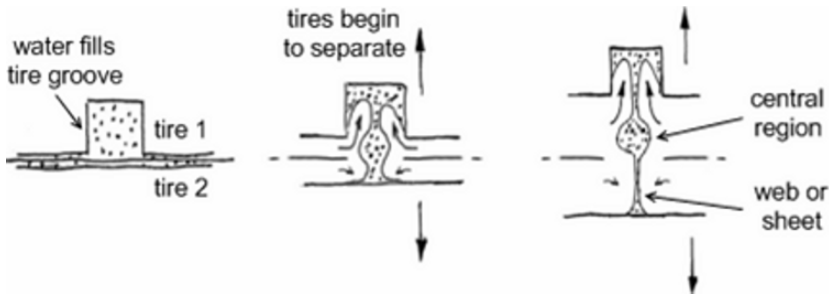


Fig. 3.2 Expected behavior of water as it leaves the tire groove at the contact patch

Eventually, ribs and sheets become a droplet field having a distribution of droplet sizes. At places where the sheet is thinnest, small droplets form along the edges of breaks and holes in the sheet. Where the broken sheet is thicker, the water within the contracting sheet is incorporated into the rib structure. Larger droplets are produced with the break-up of the periodic rib structure.

The spray patterns produced from a rolling tire are dependent on the Weber number. As described by Lefebvre [3], the development of a liquid sheet into a droplet field is governed by the fluid properties and initial velocity of the flow. Given that the groove width and properties of water are fixed, specific Weber numbers are achieved through changes in the tire speed (and jet velocity). Figure 3.3 shows images taken at $We = 1,900$ and $48,900$. At low Weber number (Fig. 3.3a), the flow field is heavily influenced by surface tension as edges of the broken sheet are rapidly pulled in towards the rib structures and little or no sheet remains. The majority of the droplet field at low We results from the break-up of the rib structures, thus, larger droplet sizes are expected. At higher Weber number (Fig. 3.3b), the flow is driven by the fluid inertia and more liquid remains in the sheets. The breakup of these thin sheets tends to produce smaller droplets.

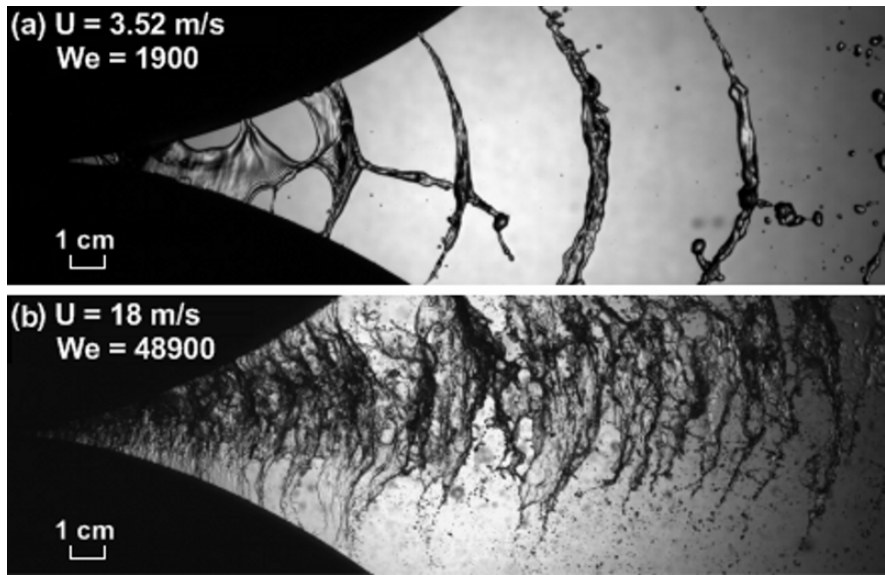


Fig. 3.3 Spray pattern variation with Weber number

3.2 Droplet Field Size and Velocity Estimates

Figure 3.4a is a single image from the video showing a portion of the droplet field produced by tires rotating at $U = 6$ m/s. The direction of flow in these images is from top to bottom. The center of the image field is one meter downstream from the end of the contact patch and the field of view is approximately 13 by 4 cm. Figure 3.4a reveals various droplet sizes as well as several “fuzzy” spots due to spray that fell outside of the contact patch at the start of the experiment. Figure 3.4b shows the gain corrected and thresholded image with only droplets in the spray plane remaining.

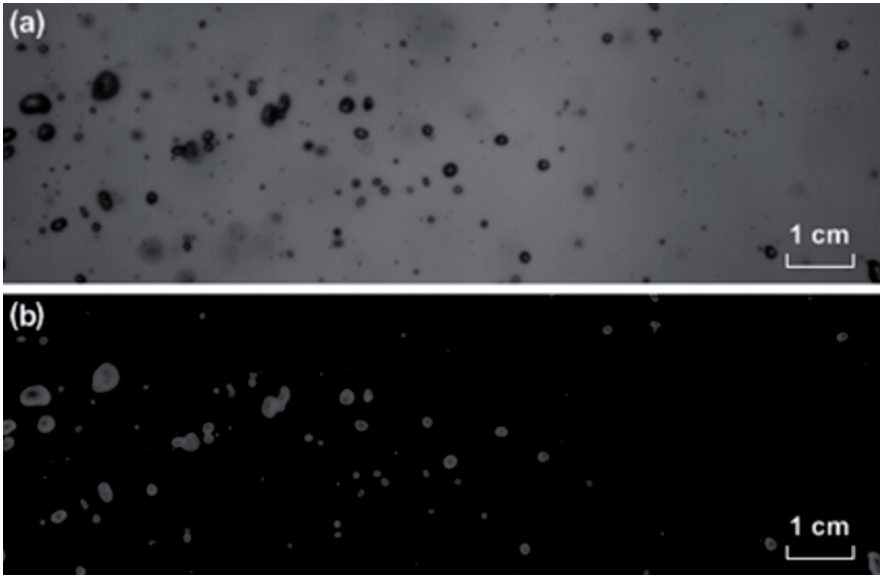


Fig. 3.4 Droplet field at $U = 6$ m/s, $We = 5,400$: (a) raw image, (b) gain corrected and thresholded image

Figure 3.5 displays the distribution of sizes for the droplets detected in this video frame. A total of fifty-eight (58) droplets were detected, ranging from 0.8 to 4.4 mm in diameter. This distribution is derived from the single image presented in Fig. 3.4. Statistically more reliable distributions can be built up by adding contributions from multiple (hundreds of) frames.

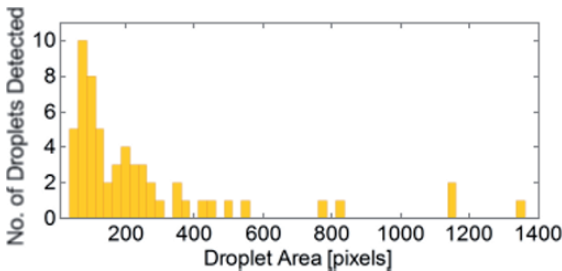


Fig. 3.5 Distribution of droplet sizes

Given a distribution of droplet diameters, images can be “filtered” for specific droplet sizes and DPIV performed to estimate droplet velocities as a function of diameter. Figure 3.6 represents a filtered version of Fig. 3.4b with only droplets from 1.65 to 2.33 mm in diameter shown. This image and a successive image taken 625 μ s later were processed using DPIV to obtain the velocity vectors for each droplet in this size range. The DPIV process tracks the motion of a droplet by

cross-correlating the two images. What makes this application of DPIV different from the standard technique is that only regions adjacent to particle centroids in the first image are interrogated in the second image. This greatly increases the efficiency of the DPIV algorithm. Typical particle displacements in the image pairs are on the order of forty pixels. The velocity estimate for droplets in this size range was 6.85 ± 0.5 m/s. The two droplets presented without a velocity vector had moved out of the image view before the second image was recorded.

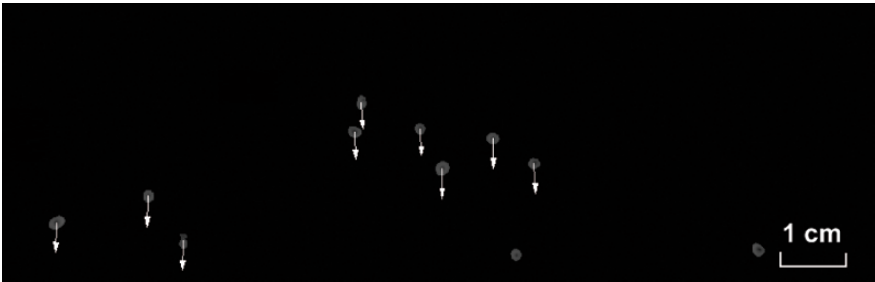


Fig. 3.6 $U = 6$ m/s spray field, filtered for droplets between 1.65 and 2.33 mm in diameter

The DPIV process was repeated for a smaller set of droplets with diameters between 1.16 and 1.65 mm. The estimated velocity for this size range was 6.15 ± 0.3 m/s. The spatial location, with respect to the contact patch centerline, and velocity vectors for these two droplet diameter ranges is presented in Fig. 3.7. A divergence away from the contact patch centerline is evident by the direction of the velocity vectors. This behavior is due to the tendency of water to attach to the surface of a tire and the ability of a tire to carry water laterally away from the plane of symmetry.

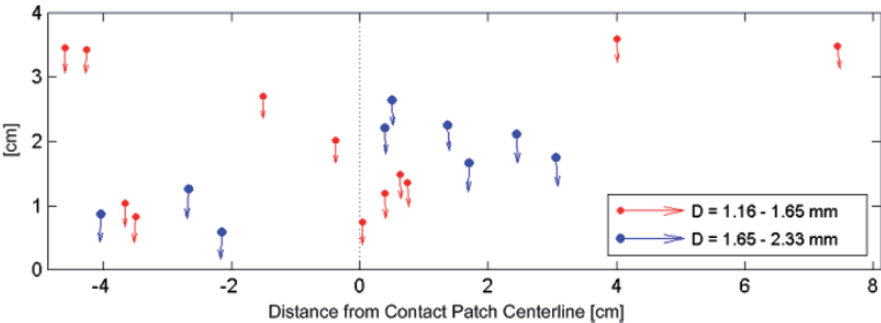


Fig. 3.7 A demonstration of DPIV results for two droplet size ranges

4 Conclusions

The operation of a tire spray simulator has been demonstrated. Video of the developing spray was captured in the region just downstream of the tire patch. From this video the flow was qualitatively described as a jet exiting the tire patch that spreads out into a sheet which spans the distance between the two tires. The sheet, thickened by the central jet, is unstable and forms sinuous waves that break into a cross-stream rib structure. Ultimately, the entire flow disintegrates into a droplet field.

A technique for isolating and measuring the size and velocity of individual droplets has been developed and calibrated. The ability to detect droplet sizes a factor of 5 different in diameter using a single exposure was demonstrated, with the lower limit of detection approaching 50 pixels in area. It is possible that this lower limit can be further relaxed.

Acknowledgments The authors would like to acknowledge the continued financial support from the Michelin Americas Research & Development Corporation as well as the production of tires for the experiment. Support from the Department of Energy (DOE) is also acknowledged. Additional thanks to Rose McCallen, Fred Browand, Jim Ross and ECI for the organization of The Aerodynamics of Heavy Vehicles II conference.

References

- [1] Marmottant P, Villermaux E, On Spray Formation, *J. Fluid Mech.*, 2004, Vol. 498, pp. 73-111.
- [2] Eastwood C D, Armi L and Lasheras J C, The Breakup of Immiscible Fluids in Turbulent Flows, *J. Fluid Mech.*, 2004, Vol. 502, pp. 309-333.
- [3] Lefebvre A, *Atomization and Sprays (Combustion: An International Series)*, CRC, 1988.

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

Florian Iser¹ and Raimund A. Almbauer²

¹ The Virtual Vehicle Competence Center, Area 2, Inffeldgasse 21a, 8010 Graz, Austria, e-mail: florian.iser@v2c2.at

² Institute for Internal Combustion Engines and Thermodynamics, Graz University of Technology, Inffeldgasse 25, 8010 Graz, Austria, e-mail: almbauer@vkmb.tugraz.at

Abstract This paper combines heavy vehicle aerodynamics with environmental considerations. The problem of air pollution in urban environments is strongly connected to the emissions of local traffic. Moreover, by-pass road tunnels in densely populated areas use often longitudinal ventilation systems, which produce high concentrations at the outlet portal. An EU-LIFE project started to investigate the potential and the feasibility of combining dust filters with standard noise protection systems. As part of the project one application deals with the reduction of pollutant emissions from road tunnel portals. For the investigation of the effectiveness of these filters, simulations of a simplified heavy vehicle shape were conducted using Computational Fluid Dynamics (CFD). The first part of this paper deals with the validation of the CFD tool. Therefore, the Ground Transportation System (GTS) in the NASA Ames 7 feet by 10 feet wind tunnel was simulated and the results were compared to measurements. The second part covers the considerations of the flow field induced by heavy vehicle traffic within a highway road tunnel equipped with filters. As a result, the amount of the volume flow through the filters for one complete heavy vehicle cycle is displayed.

1 Introduction

Many cities in the world face poor air quality in their urban environments. Especially the European Union supports activities to reduce the amount of very fine particles, named particulate matter. The smaller the particles are the higher is the risk of serious impact on human organs. Therefore the World Healthy Organization (WHO), the European Union (EU), and federal governments released limits for the maximum concentration of particles smaller than 10 μm (PM10) in diameter.

For the member states of the European Union an exceedance of the daily mean threshold value of $50 \mu\text{g}/\text{m}^3$ of less than 35 days per year is allowed at the present time. The European Acid Deposition model system (EURAD) [5] predicts the concentration of PM10 over Europe. Especially, on winter days, large areas of Europe exceeded the daily mean limit. For example, the city of Klagenfurt which is located in the south of Austria, exceeded the daily mean limit on 80 days during the winter season between October and March. That means, nearly every second day the concentration of PM10 exceeded the threshold values.

On the other hand, the issue of noise pollution from traffic is becoming more important. In the year 2006, more than EUR 85 million was invested in Austria to build noise protection walls near highways. Currently, about 430 km of the 2060 km long Austrian highway network is provided with noise barriers. Furthermore, 20 % to 30 % of the total costs for new highway constructions are invested into noise protection systems [1].

The Virtual Vehicle Competence Center and the Graz University of Technology started a project together with five other partners to investigate the feasibility to reduce the PM10 concentration from traffic sources where they originate. The Sound and Particle Absorbing System (SPAS) should be able to combine the functions of a standard noise barrier with an integrated dust filter near streets. This paper deals with the road tunnel application of the SPAS elements. The target is to reduce the PM10 emissions at the tunnel outlet portal with no additional effort than the flow energy of the traffic.

1.1 PM10 Sources of Klagenfurt

Based on a previous project, named KAPA GS, accomplished by the Graz University of Technology and regional partners in 2005, the primary sources of PM10 emissions for the city of Klagenfurt were determined. As a result, 52 % of the PM10 emissions are generated by local traffic. However, only one-third of the traffic emissions are exhaust emissions whereas the remaining two-third are non-exhaust emissions like brake wear, tire wear, street wear, and resuspension.

1.2 Road Tunnel Situation

Inside a road tunnel higher concentrations of pollutants occur because of a reduced dilution with fresh air compared to non-tunnel situations. Especially for self-ventilated road tunnels which have no intermediate air exchange all traffic emissions are transported to the outlet portal. Results of different dispersion

models by Oettl et al. [7, 8] shows the high concentration of polluted air in the environment due to emissions at the tunnel end. Even for ventilated road tunnels dispersion of traffic pollutants is a big issue.

An advantage for the CFD calculation of a road tunnel is the constant and well known boundary conditions. The geometry data of the tunnel cross-section is known, the traffic speed is nearly constant and after a few hundred meters, inside the tunnel climate influences are strongly reduced. Even under snowy conditions the tunnel end is usually dry. Nevertheless, there are also two critical points concerning the tunnel application which have to be mentioned. One is the clearance issue of the tunnel geometry. Every additional mounted device must be within the clearance profile. The second point is the various safety regulations. The SPAS elements must fulfill all fire-retarding requirements as well as all durability considerations.

The moving traffic inside an unidirectional road tunnel generates a vehicle-induced constant velocity. The force contributed by each vehicle depends on the size, its blockage ratio [4] and various other parameters such as traffic frequencies and pressure difference at the tunnel portals. The SPAS filters for this tunnel consideration are intended to be placed at each side wall and additionally on the tunnel ceiling. To provide enough space for the flow, a distance of 100 mm was considered between the filters backside and the tunnel walls. This configuration was investigated and the volume flow through each filter was calculated. Moreover, a volume rate per unit length was derived with a constant traffic speed of 23 m/s.

To obtain validated CFD settings, the following chapter describes the preceded calculation with the GTS. Later in Section 3, the CFD setup was transferred to the real tunnel geometry with an adapted GTS. Finally, the volume flow values for the SPAS filters were evaluated.

2 Preceded CFD Calculations

This section describes the CFD calculations which were conducted prior to the tunnel application. The target was to validate the CFD calculation for a similar geometrical setup with wind tunnel measurements. Furthermore, this preceded CFD simulation validates the simulation method, especially concerning the mesh and turbulence modeling.

2.1 Problem Formulation

For the use of a generic shape for heavy vehicles the Ground Transportation System (GTS) was chosen which was introduced by Gutierrez et al. [2]. The GTS shape is a simplified tractor-trailer configuration using a cab-over tractor design. A 1/8 scale model of the GTS was measured at the NASA Ames 7 feet by 10 feet wind tunnel [13]. All length parameters are non-dimensionalized by the trailer width $w = 0.3238$ m. To reach a Reynolds number that is comparable to the full size case, high velocities inside the test section of more than 90 m/s were necessary to obtain $Re = 2,000,000$. However, Mach numbers of about 0.28 demand compressible flow simulation. The basis for the CFD validation presented herein was the NASA experiment at run 7 point 9.

2.2 Numerical Setup

All CFD calculations were conducted with CD-adapco's STAR-CCM+ V2. This new developed commercial code offers a fully integrated automatic mesh tool which provides a surface wrapper and a hybrid mesh generator. For the GTS calculations a trimmed mesh was deployed which is an unstructured hexahedral mesh with trimmed cells near the wall. To make it comparable with CFD calculations of Roy et al. [11] and to avoid validation issues concerning the wall treatment, the boundary layer was fully resolved. Hence, the non-dimensional wall distance was $y^+ \leq 0.5$. Menter's BSL $k-\omega$ turbulence model was used, because this configuration delivered the best results for the GTS model, as already published by various authors [9, 11, 12]. Refinement regions were provided at the trailer base and in the gap between the GTS bottom and the wind tunnel floor. For the wall boundaries a number of 30 prism cells parallel to the surface were employed. Furthermore, the mesh size on the GTS surface varied from 2 mm up to 8 mm. These mesh settings lead to an overall number of 21.8 million cells. Finally, the boundary setup for the calculation including the GTS is shown in Fig. 2.1.

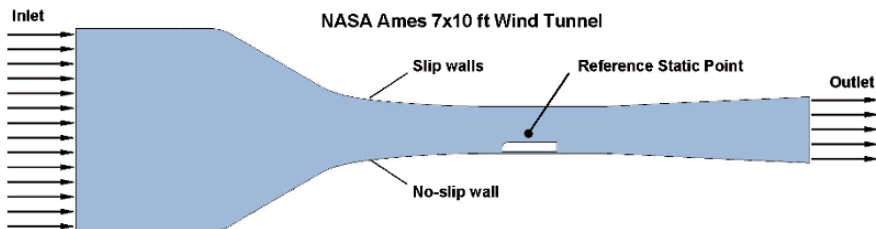


Fig. 2.1 Boundary setup for the NASA Ames wind tunnel including the GTS

2.3 Results

The first consideration was to validate the boundary layer profile of the floor for an empty wind tunnel configuration. Figure 2.2 compares the profile of the CFD calculation with the ones based on the NASA experimental data of run 404. The CFD calculation provided a very similar profile compared to measured data and is used for further investigations.

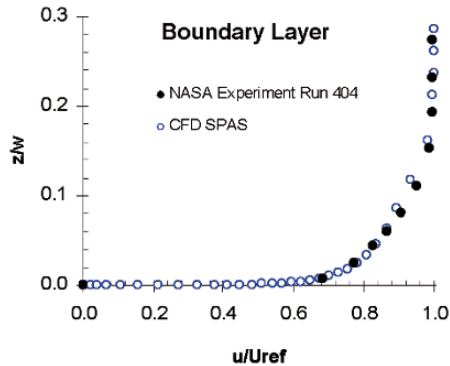


Fig. 2.2 Comparison of measured and calculated boundary profile for an empty NASA Ames wind tunnel

The GTS calculations were run in parallel on an Opteron cluster using 24 CPUs. The first detailed discussion of the results is dealing with the pressure distribution on the surface. Furthermore, Fig. 2.3 gives the surface pressure results for the front and base region of the GTS. It can be seen that the distribution on the front end conforms to measurements. However, at the trailer base, only the pressure level is predicted correctly but not the pressure distribution.

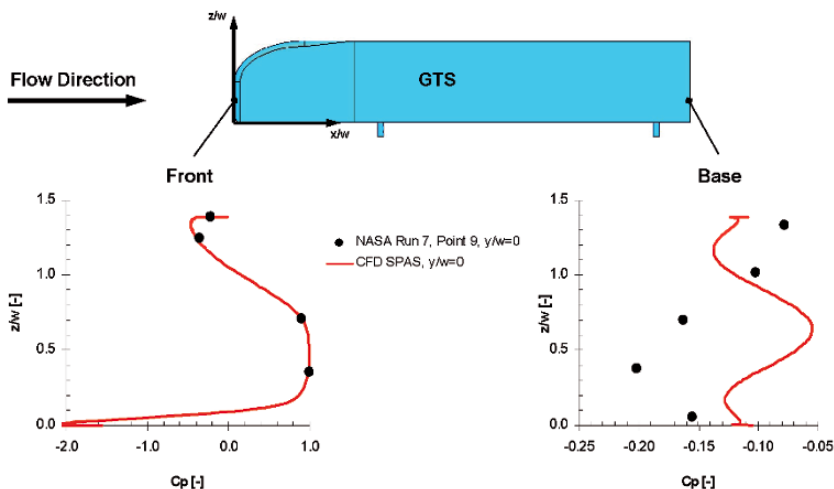


Fig. 2.3 Comparison of measured and calculated surface pressures on the front and base of the GTS

To evaluate the reason for this characteristic, streamlines were calculated and compared to particle image velocimetry (PIV) measurements which were also contributed by the NASA experiment. The trailer wake in the horizontal cut shows a similar profile between the CFD results and the PIV data (Fig. 2.4). Nevertheless, the vertical cut displays the reason for the surface pressure discrepancy. So, the shape of the trailer wake could not be predicted in a sufficient way by the turbulence model. These findings were already discussed by Roy et al. [11].

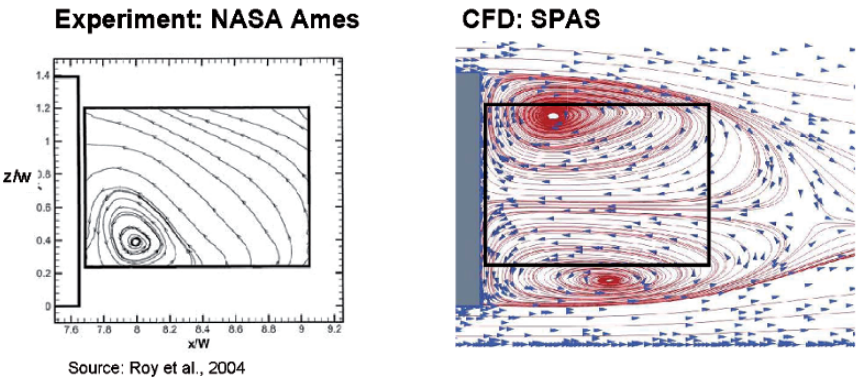


Fig. 2.4 Comparison of measured [10] and calculated streamlines for the horizontal cross-section in the middle of the GTS base

The integration of the surface pressure field results in the drag coefficient C_D . As a result, the drag coefficient for the GTS without the four posts was calculated

to $C_D = 0.237$, which is 5.4 % below the measurement value. Table 2.1 compares the experimental data with the different CFD results.

Table 2.1 Drag coefficient for the GTS from various sources

	C_D	Error	Source
NASA Ames experiment	0.250 ± 0.01		[11]
Menters k- ω	0.237	5.4 %	
Menters k- ω (Roy et al.)	0.298 ± 0.06	19.2 %	[13]

This presented CFD setup was used for the prediction of the flow field around SPAS filters inside the road tunnel, which is described in the following chapter.

3 Road Tunnel Application

This section deals with the application of the SPAS elements in a road tunnel and with the transformation of the CFD setup for a predictive task. The first section deals with the provided heavy vehicle shape, whereas the following sections describe the tunnel setup and the CFD calculations, respectively.

3.1 Ground Transportation System for the EU

The use of a generic heavy vehicle shape is reasonable because of the cab-over design of European truck manufactures. To meet the requirements of the European legislation, the overall heavy vehicle length is limited to 16.5 m, with a width of 2.5 m, and a height of 4 m. The original GTS's width is 2.6 m, so it was scaled down to 2.5 m and automatically the height of 3.96 m was reached. However, the overall length was too long; therefore a section of the trailer was cut out to obtain the 16.5 m in total length. This modified shape is referred in this paper as the EU-GTS for obvious reasons.

3.2 Road Tunnel Setup

The chosen road tunnel, named according to the nearest village Lendorf, is located at the highway A2 in the area of the city of Klagenfurt. The shape of the tunnel cross-section is rectangular and the total length is about 800 m. The detailed tunnel information is summed up in Table 3.1. The tunnel is self-ventilated

and therefore high pollution concentrations, including PM10, can be expected at the tunnel outlet portal. Figure 3.1 shows the arrangement of the EU-GTS inside the tunnel. The coordinate system is equal to the preceded GTS calculation described in Section 2. To avoid interactions of the tunnel portal because of the changing geometry, the calculated section does not include any tunnel end.

Table 3.1 Detailed information for the road tunnel Lendorf

Caption	Value
Daily traffic (for each direction, counted 2003)	15,500 vehicles
Cars-to-heavy vehicles ratio (fleet composition, counted 2003)	83 % to 17 %
Total tunnel length	800 m
Tunnel cross-sectional area	47.5 m ²
Longitudinal velocity (measured in similar tunnels)	7 m/s
Exhaust PM10 emission for the fleet	0.026 g/km
Non-exhaust PM10 emission for the fleet	0.052 g/km

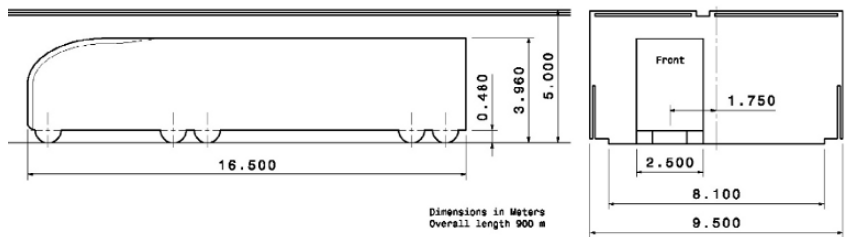


Fig. 3.1 Arrangement of the EU-GTS inside the road tunnel Lendorf

3.3 Computational Setup

The computational domain was very similar to those of the GTS in the NASA Ames wind tunnel. However, the tunnel walls were set as no-slip walls and additional refinements were necessary to obtain a higher resolution between the SPAS filters and the tunnel walls. As described in section 3.1 the EU-GTS required the same refinement regions as the GTS which is known from the preceded simulation.

Nevertheless, some additional settings were taken embedding the SPAS filter material:

- Porosities for the SPAS filters cells
- Translational Moving Reference Frame (MRF) for the SPAS filters cells
- Higher inlet turbulence due to traffic inside the tunnel
- Incompressible flow
- Modified cell sizes

The porosity values for the filters were measured at the Virtual Vehicle's own wind tunnel. Several materials were investigated to obtain a low pressure loss. To consider the stationary CFD simulation, the EU-GTS stands still and the tunnel walls are moved with a surface velocity of the vehicle speed. However, due to the porosity values, the filter cells themselves allow only a flow direction normal to the surface (y-axis) and therefore they need an additional reference velocity component (x-axis). These settings were made with a translational MRF approach. Furthermore, all end faces of the four filters were connected with the starting faces by using a periodic boundary condition.

For the amount of turbulence at the tunnel inlet the investigation of Lindener et al. [6] shows turbulent intensities and turbulent length scales for heavy traffic on highways. These measurements were taken from non-tunnel situations and provide a guide value. In the work of Hall [3], who dealt with gas dispersion in road tunnels, the turbulence intensity and the length scale were given within 10 % and 10 % of the tunnel diameter, respectively. As a result, a turbulence intensity $I = 10\%$ and a turbulent length scale $L = 0.5$ m were set for the road tunnel simulation.

The full-scale CFD simulation with a vehicle speed of 23 m/s allowed the application of the incompressible flow assumption. In order to consider a wind speed of 7 m/s in the longitudinal direction of the tunnel due to traffic drag, the intake flow velocity was reduced to 16 m/s to generate a resulting vehicle speed of 23 m/s for the simulation. Thus, the longitudinal velocity generates a boundary layer on the walls. Therefore, all wall boundaries are resolved by 30 parallel cell layers. To provide similar y^+ -values for the EU-GTS as for the GTS, the cell sizes were adjusted. For example, the wall distance for first boundary layer near the surface was set to 0.016 mm. The modifications of the mesh lead to 22.7 million cells and a y^+ of approximately 0.8 for the EU-GTS surface, which was important to fulfill the requirements of the low- y^+ turbulence model.

3.4 Results

The simulation of the EU-GTS inside the road tunnel shows the impact of the high blockage ratio. The calculation was run on 26 CPUs in parallel for 178 hours. In fact, a convergent solution was reached within 6500 iterations. Again, the drag coefficient for the EU-GTS without the posts was calculated to $C_D = 0.293$ for this road tunnel setup. Because of the missing reference point, a direct comparison of the surface pressure distribution between the GTS and the EU-GTS was not possible.

All length values presented in this section are non-dimensionalized with the EU-GTS width $W = 2.5$ m. In Fig. 3.2 to 3.4, three cross-sections along the x-axis provide an understanding of the interaction between the flow field and the EU-GTS even far away from the vehicle shape. The cuts at $x/W = 0.10$, $x/W = 1.50$, and $x/W = 6.68$ display the positions at the front of the EU-GTS, in the middle of the trailer, and immediately behind the rear end, respectively. As a result, the tangential velocity magnitude v_t represents the displacement due to the EU-GTS body. Furthermore, the position of the SPAS filters can be adopted due to these results.

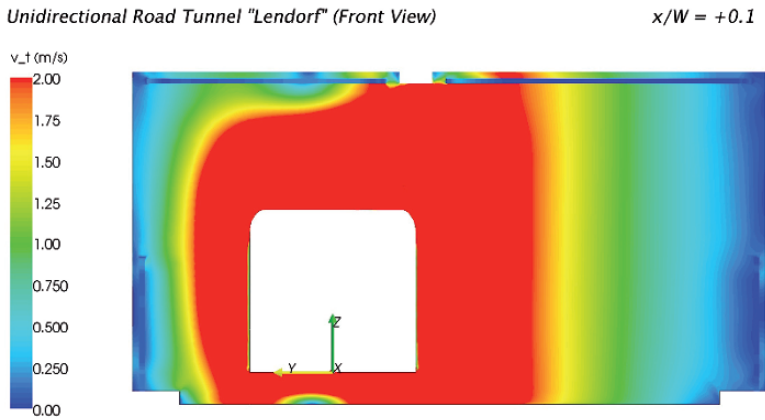


Fig. 3.2 Tangential velocity magnitude of a vertical plane cut at $x/W = 0.1$ at the EU-GTS front

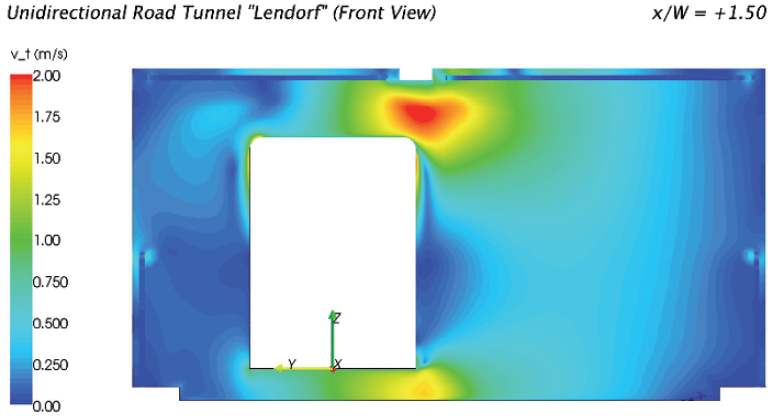


Fig. 3.3 Tangential velocity magnitude of a vertical plane cut at $x/W = 1.5$ at the middle of trailer

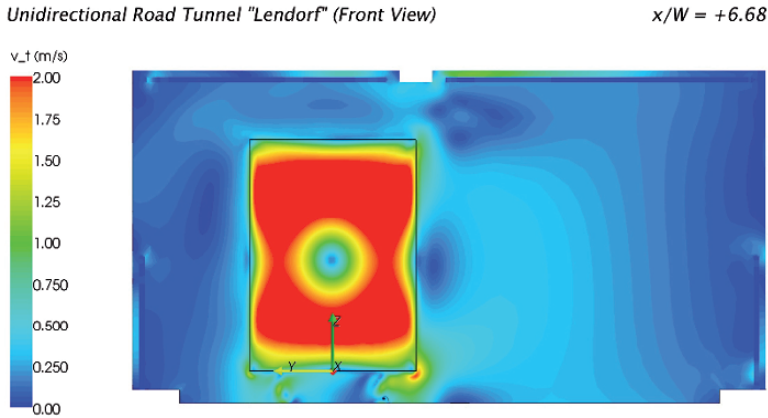


Fig. 3.4 Tangential velocity magnitude of a vertical plane cut at $x/W = 6.68$ immediately behind the trailer base

Figure 3.5 shows the velocity v_y which is normal to the SPAS filter surface on a horizontal line. This line is drawn at a height of $z/W = 0.4$ inside the SPAS filter cells of filter A. The velocity v_y begins and ends at rest; therefore one complete heavy vehicle cycle is captured by the plot. However, v_y depends strongly on the z/W -position.

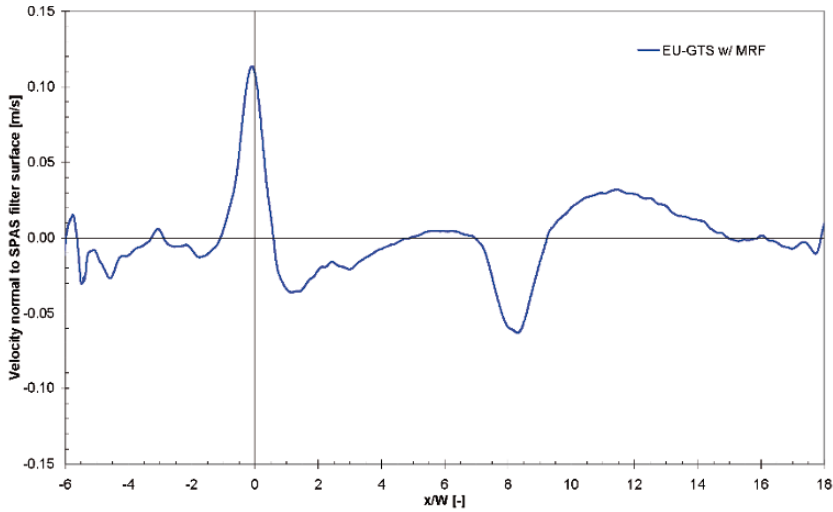


Fig. 3.5 Velocity v_y normal through SPAS filter A on a line-probe at $z/W = 0.4$

For the prediction of the entire volume flow through the SPAS filters, the product of filter area and v_y have to be integrated. Furthermore, the volume flow was separated into forward and backward direction to obtain an in-flow and out-flow value. The notation of all four SPAS filters as well as the directions for in- and out-flows are shown in Fig. 3.6. To obtain the volume for 1 m length of all four SPAS filters, the volume flow must be multiplied by the time per meter length, which is the reciprocal value of the vehicle speed (Table 3.2). These calculated values are valid for only one EU-GTS and must be multiplied with the heavy vehicle traffic.

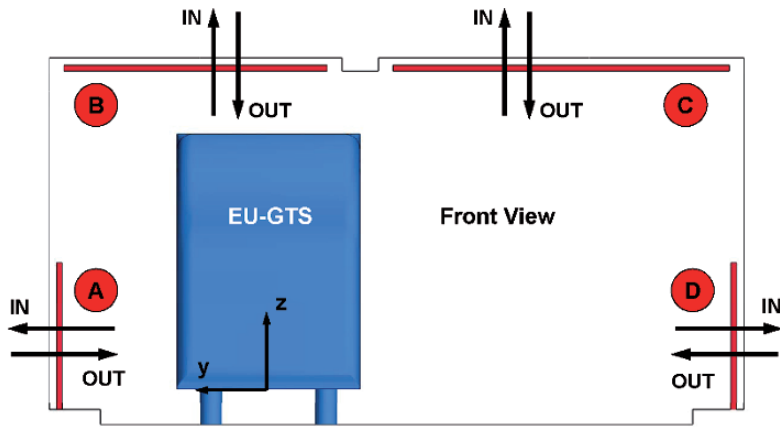


Fig. 3.6 SPAS filter caption and in-flow, out-flow definition

Table 3.2 Volume flow and volume per unit length through the SPAS filters

SPAS Filter	V'_{in} (m ³ /s)	V'_{out} (m ³ /s)	V_{in}/l (m ³ /m)	V_{out}/l (m ³ /m)
A	1.454	1.257	0.0632	0.0546
B	1.185	0.106	0.0515	0.0046
C	2.983	1.224	0.1297	0.0532
D	1.488	1.096	0.0647	0.0476

4 Conclusions and Outlook

Local traffic contributes a significant part to bad air quality in cities. The target of the EU-LIFE SPAS project is a reduction of traffic emissions where they originate, and in case of a road tunnel this is at their outlet portals. Therefore, filters were investigated to evaluation the volume flow which can be later used to estimate the effectiveness of dust filtration systems.

To obtain the flow field around such SPAS filter elements, a simplified shape of a European-like heavy vehicle was primarily simulated inside the NASA Ames wind tunnel to validate the CFD-code and the appropriate settings. Then these findings were transferred to a self-ventilated road tunnel, where the SPAS filters were mounted on the tunnel walls as well as on the ceiling. As a result, the volume per length which flows through the SPAS filters delivered an order of magnitude for the further filtration effort, especially PM10.

Further CFD simulations will be dealing with the influence of the exact position of heavy vehicles driving through road tunnels. A previous parameter study, which was made under different conditions, provided a strong non-linear influence of the distance between the heavy vehicle and the tunnel wall. Moreover, detailed models of European heavy vehicles will result in more complex flow fields inside a road tunnel and will help to obtain a better understanding of the flow mechanisms.

Acknowledgments

The authors would like to thank Bruce Storms from NASA Ames for the very helpful data of the experimental investigation of the GTS in the NASA Ames wind tunnel. This project is funded by the European Commission as part of the EU-LIFE project.

References

- [1] Autobahnen- und Schnellstraßen-Finanzierungs-Aktiengesellschaft: Annual Report of the ASFINAG Group (2006)
- [2] Gutierrez W, Hassan B, Croll R, Rutledge W: Aerodynamics Overview of the Ground Transportation Systems (GTS) Project for Heavy Vehicle Drag Reduction. SAE Paper (960906 SP-1145) (1996)
- [3] Hall R: Modelling of Dense Gas Dispersion in Tunnels. Contract research report 359/2001, HSE -Health and Safety Executive (2001)
- [4] Hucho W (ed.): Aerodynamics of Road Vehicles, 4th edn. SAE, Warrendale (1998)
- [5] Langmann B, Graf H: The Chemistry of the Polluted Atmosphere Over Europe: Simulations and Sensitivity Studies with a Regional Chemistry-Transport-Model. *Atmospheric Environment* 31(19), 3239–3257 (1997)
- [6] Lindener N, Miehl H, Cogotti A, Cogotti F, Maffei M: Aeroacoustic Measurements in Turbulent Flow on the Road and in the Wind Tunnel. In: J. Wiedemann (ed.) *Progress in Vehicle Aerodynamics and Thermal Management V*, pp. 193–211. FKFS -Research Institute of Automotive Engineering and Vehicle Engines Stuttgart (2007)
- [7] Oetl D, Sturm P, Almbauer R, Okamoto S, Horiuchi K: Dispersion from Road Tunnel Portals: Comparison of two Different Modelling Approaches. *Atmospheric Environment* 37, 5165–5175 (2003)
- [8] Oetl D, Sturm P, Bacher M, Pretterhofer G, Almbauer R: A Simple Model for the Dispersion of Pollutants from a Road Tunnel Portal. *Atmospheric Environment* 36, 2943–2953 (2002)
- [9] Pointer W, Sofu T, Weber D: Development of Guidelines for the Use of Commercial CFD in Tractor-Trailer Aerodynamic Design. SAE (SAE 2005-01-3513) (2005)
- [10] Roy C, Payne J, McWherter-Payne M, Salari K: RANS Simulations of a Simplified Tractor/Trailer Geometry. In: R. McCallen, F. Browand, J. Ross (eds.) *Lecture Notes in Applied and Computational Mechanics -The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains*, vol. 19, pp. 207–218. Springer (2004)
- [11] Roy C, Payne J, McWherter-Payne M: RANS Simulations of a Simplified Tractor/Trailer Geometry. *Journal of Fluids Engineering* 128, 1083–1089 (2006)
- [12] Salari K, Ortega J, Castellucci P: Computational Prediction of Aerodynamic Forces for a Simplified Integrated Tractor-Trailer Geometry. AIAA Paper (AIAA-2004-2253) (2004)
- [13] Storms B, Ross J, Heineck J, Walker S, Driver D, Zilliac G: An Experimental Study of the Ground Transportation System (GTS) Model in the NASA Ames 7-by 10-Ft Wind Tunnel. NASA-TM (2001-209621) (2001)

Experiments and Calculations Relevant to Aerodynamic Effects during Highway Passing Maneuvers

B. Basara¹, S. Girimaji², S. Jakirlic³, F. Aldudak³ and M. Schreffl⁴

¹ AVL List GmbH AVL List GmbH, Hans List Platz 1, 8020 Graz, Austria

² Aerospace Engineering Department, Texas A&M University, TX 77843, USA

³ Fluid Mechanics and Aerodynamics, Darmstadt University of Technology, Germany

⁴ BMW AG, Aerodynamics, Munich, Germany

branislav.basara@avl.com

Abstract Aerodynamic effects during highway passing maneuvers are still not well documented. To better understand these effects, a 40% car-truck overtaking process has been carried out in the BMW wind tunnel. As a first step, the car aerodynamics has been measured without the truck to establish the reference pressure distribution for subsequent tests. The overtaking process has been approximated by fixing the truck model at eight stationary positions relative to the car model. This approximates the overtaking process as a quasi-stationary process. The reference calculations are performed with a new variant of $\overline{v^2} - f$ model, the so-called ζ -f model (Hanjalic et al., 2004). Furthermore, the calculations are also performed by using recently proposed Partially-averaged Navier Stokes (PANS) model (Girimaji et al., 2003, Girimaji and Hamid, 2005) in order to capture unsteady effects more correctly compared with the unsteady RANS. Because of its extensive reference database, the well-known Ahmed body benchmark (25 degrees) was computationally investigated as an introductory case with respect to the comparative assessment of the RANS and PANS approaches. A validation procedure for the PANS method is then conducted by computing the single passenger car (40% BMW model).

Introduction

There are a number of publications in the field of the external car aerodynamics, e.g. Baxendale et al. (1996), Makowsky and Kim (2000), Basara et al. (2001) etc. However, we will highlight only two, which are the most relevant for the work presented here. In the work of Basara and Tibaut (2002), the error of the steady state approach for the flows where transient effects are important, was analyzed. And in the work of Khalighi et al. (2001), the $\overline{v^2} - f$ (Durbin, 1991) was successfully used to predict the unsteady wake behind a bluff body.

This complementary experimental and computational study focuses on turbulence model validation in such a complex flow situation. The model applied relies on the elliptic relaxation (ER) concept providing a continuous modification of the homogeneous pressure-strain process as the wall is approached to satisfy the wall conditions, thus avoiding the need for any wall topography parameter. This model approach represents a further contribution towards more robust use of advanced closure models. The variable ζ represents the ratio $\overline{v^2}/k$ ($\overline{v^2}$ is a scalar property in the Durbin's $\overline{v^2} - f$ model, which reduces to the wall-normal stress in the near-wall region) providing more convenient formulation of the equation for ζ and especially of the wall boundary conditions for the elliptic function f . This model is used in conjunction with the so-called universal wall treatment. The latter method blends the integration up to the wall (exact boundary conditions) with the wall functions, enabling well-defined boundary conditions irrespective of the position of the wall-closest computational node. This method is especially attractive for computations of industrial flows in complex domains where higher grid flexibility, i.e. weaker sensitivity against grid non-uniformities in the near wall regions, featured by different mean flow and turbulence phenomena (flow acceleration/deceleration, streamline curvature effects, separation, etc.), is desirable. All simulations were performed by using the commercial CFD software package AVL FIRE v8.5 (2006).

The corresponding wind tunnel tests were performed in the full scale wind tunnel of the BMW Group in Munich. Both, tests on 1:2.5 scale models and on a full-scale cars were performed. Despite geometrical differences, the data obtained for both car configurations can be used for mutual comparison. The aerodynamic effects during highway passing maneuvers are still not completely understood. To better understand these effects, a 40% overtaking process has been planned in the BMW wind tunnel, Fig. 2. As a first step, the car aerodynamics has been measured without the truck to gain the reference pressure distribution for subsequent tests, Fig. 1. This simplification to a 40% model is acceptable and no significant discrepancies to full scale pressure field are to be expected. This is in accordance with Széchényi (2004), Yamamoto et al. (1996). This simplified configuration has been chosen to be capable to extend the measurements in a second step to overtaking processes which have been approximated by eight stationary relative positions. By doing so the unsteady overtaking process has been simplified to a set of

stationary process. In the present work only the flow past a single passenger car configuration (40% model) was considered.

Experimental Details

The 40%-model was equipped with 54 surface-mounted pressure tabs: 13 pressure tabs were mounted on each car-side; 14 on bonnet, roof and trunk, see Figs. 1-3 for their distribution at the car surface. In addition, the stagnation point pressure has been acquired. The pressure tabs were wired to the car interior and connected to a 64-channel PSI pressure module to measure the surface pressure distribution in reference to the plenum pressure. The car model was a foam model. Tests have been performed at a velocity of 140 km/h, resulting in a Reynolds-number of 9×10^5 .



Fig. 1 Wind tunnel configuration



Fig. 2 Wind tunnel configuration in the case of overtaking process

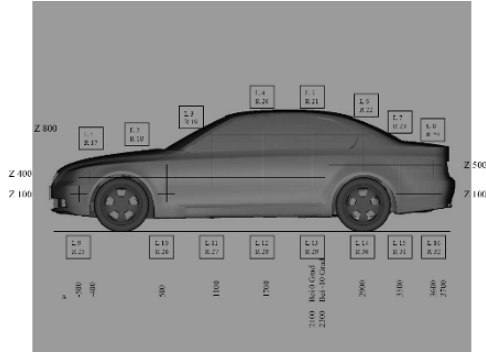


Fig. 3 Pressure tabs distribution at the side surface

Computational Model

The flow field is modeled by the Reynolds-Averaged Navier Stokes equations coupled with the eddy-viscosity $\zeta - f$ model equations. The continuity and momentum equations may be written as:

$$\frac{\partial}{\partial x_j}(\rho U_j) = 0 \quad (1)$$

$$\frac{\partial}{\partial t}(\rho U_i) + \frac{\partial}{\partial x_j}(\rho U_i U_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\tau_{ij} - \rho \overline{u_i u_j}) \quad (2)$$

where ρ is the fluid density, U_i is the mean-velocity vector, p is the static pressure, t is time and τ_{ij} denotes the mean viscous stress tensor:

$$\tau_{ij} = 2\mu S_{ij} \quad (3)$$

In the above, μ is the dynamic viscosity and the mean strain rate tensor S_{ij} is given as:

$$S_{ij} = \frac{1}{2} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (4)$$

The last term in Equation (2) ($-\rho \overline{u_i u_j}$) is the unknown Reynolds stress tensor which is obtained from the constitutive relation analogous to Equation (3) (Boussinesq's analogy):

$$-\rho \overline{u_i u_j} = 2\rho \nu_t S_{ij} - \frac{2}{3} \rho k \delta_{ij} \quad (5)$$

where μ_t is the turbulent viscosity, δ_{ij} is the Kronecker delta and $k = 0.5 \overline{u_i u_i}$ represents the turbulent kinetic energy. In the $\zeta - f$ (Hanjalic et al. 2004) model the eddy-viscosity is defined as:

$$\nu_t = C_\mu \zeta k \tau \quad (6)$$

where ζ is the velocity scale ratio obtained from the following equation:

$$\frac{D\zeta}{Dt} = f - \frac{\zeta}{k} P_k + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_\zeta} \right) \frac{\partial \zeta}{\partial x_j} \right] \quad (7)$$

The time scale τ is given as

$$\tau = \max \left[\min \left(\frac{k}{\varepsilon}, \frac{a}{\sqrt{6} C_\mu S \zeta} \right), C_\tau \left(\frac{\nu}{\varepsilon} \right)^{1/2} \right] \quad (8)$$

The coefficients appearing in Equations (6) and (8) take the values $\alpha = 0.6$,

$C_\mu = 0.22$ and $C_\tau = 6.0$. The complete model is given by following equations:

$$\frac{Dk}{Dt} = (P_k - \varepsilon) + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \quad (9)$$

$$\frac{D\varepsilon}{Dt} = \frac{C_{\varepsilon 1} P_k - C_{\varepsilon 2} \varepsilon}{\tau} + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] \quad (10)$$

In above equations, the production of k is given as:

$$P_k = -\overline{u_i u_j} \frac{\partial U_i}{\partial x_j} \quad (11)$$

These equations are solved in conjunction with an equation for the elliptic relaxation function f which is formulated by using the pressure-strain model of Speziale, Sarkar and Gatski (1991) and given as:

$$L^2 \nabla^2 f - f = \frac{1}{\tau} \left(c_1 + C_2' \frac{P_k}{\varepsilon} \right) \left(\zeta - \frac{2}{3} \right) - \left(\frac{C_4}{3} - C_5 \right) \frac{P_k}{k} \quad (12)$$

where f goes to zero at the wall: $f_{wall} = \lim_{y \rightarrow 0} \frac{-2\nu\zeta}{y^2}$ and the length scale L is obtained from:

$$L = C_L \max \left(\min \left(\frac{k^{3/2}}{\varepsilon}, \frac{k^{1/2}}{\sqrt{6}C_\mu S\zeta} \right), C_\eta \left(\frac{\nu^3}{\varepsilon} \right)^{1/4} \right) \quad (13)$$

Hanjalic et al. (2004) neglected the last term in Eq. (12) due to small values of $(C_4/3 - C_5) \approx 0.008$, and they also decreased C_2' from the original SSG value 0.9 to 0.65 in order to take into account the discrepancy in the definition of ε in the log-law region. The $\zeta - f$ model is very robust and more accurate than the simpler two-equation eddy viscosity models (Hanjalic et al., 2004). Nevertheless, the model is usable for a relatively coarse mesh next to the wall but again the cell next to the wall should reach a non-dimensional wall distance y^+ as a maximum less than 3. The largest error is introduced when the first computational cell fails in the buffer region, e.g. $5 \leq y^+ < 30$. Some authors have combined the integration up to the wall with wall functions. These smoothing functions which blend two formulations together are known as automatic wall treatment, hybrid or compound wall treatment, etc. Popovac and Hanjalic (2005) proposed the blending formula for the quantities specified at the cell P next to the wall as

$$\phi_P = \phi_\nu e^{-\Gamma} + \phi_t e^{-1/\Gamma} \quad (14)$$

where ' ν ' is the viscous and ' t ' the fully turbulent value of the variables: wall shear stress, production and dissipation of the turbulence kinetic energy. Their compound wall treatment was introduced under the name "Hybrid Wall Treatment" in AVL FIRE due to some simplifications of the original approach. The original compound wall treatment of Popovac & Hanjalic (2005) includes the tangential pressure gradient and convection, but here, the simplest approach is used which includes the standard wall function. Another departure from the original formulation is in the calculation of the dissipation rate as proposed by Basara (2006).

The Partially-Averaged Navier-Stokes (PANS) approach is a recently proposed method by Girimaji et al. (2003), which changes seamlessly from RANS to the direct numerical solution of the Navier-Stokes equations (DNS) as the unresolved-to-total ratios of kinetic energy and dissipation are varied. There are two variants of the PANS model derived up to now, one based on the $k-\varepsilon$ formulation and another based on the $k-\omega$ formulation (Lakshmipathy & Girimaji, 2006). We have used here the first formulation where the unresolved kinetic energy and dissipation

equations are systematically derived from the k - ε model. The model equations for the unresolved kinetic energy k_u and the unresolved dissipation ε_u are given as

$$\frac{Dk_u}{Dt} = (P_u - \varepsilon_u) + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_u}{\sigma_{ku}} \right) \frac{\partial k_u}{\partial x_j} \right] \quad (15)$$

$$\frac{D\varepsilon_u}{Dt} = C_{\varepsilon 1} P_u \frac{\varepsilon_u}{k_u} - C_{\varepsilon 2}^* \frac{\varepsilon_u^2}{k_u} + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_u}{\sigma_{\varepsilon u}} \right) \frac{\partial \varepsilon_u}{\partial x_j} \right] \quad (16)$$

where the eddy viscosity of unresolved scales is equal to

$$\nu_u = c_\mu \frac{k_u^2}{\varepsilon_u} \quad (17)$$

The model coefficients are

$$C_{\varepsilon 2}^* = C_{\varepsilon 1} + \frac{f_k}{f_\varepsilon} (C_{\varepsilon 2} - C_{\varepsilon 1}); \quad \sigma_{k,\varepsilon u} = \sigma_{k,\varepsilon} \frac{f_k^2}{f_\varepsilon} \quad (18)$$

The parameter which determines the unresolved-to-total kinetic energy ratio f_k is defined based on the grid spacing, following Girimaji & Hamid (2005), thus

$$f_k = \frac{1}{c_\mu} \left(\frac{\Delta}{\Lambda} \right)^{2/3} \quad (19)$$

where Δ is the grid cell dimension and Λ is the Taylor scale of turbulence. The PANS asymptotic behaviour goes smoothly from RANS to DNS with decreasing f_k . In the work presented here, the parameter f_k is implemented in the computational procedure as a dynamic parameter, changing at each point at the end of every time step, and then it is used as a fixed value at the same location during the next time step.

Results and Discussion

The first case used for comparisons is the Ahmed Body (Ahmed, 1984), a three-dimensional idealized vehicle model which represents the key benchmark for validation of the turbulence models regarding external car aerodynamics. This case allows investigation of the back slant effect on the overall drag force. At a certain angle of the slant, a vortex breakdown phenomenon appears causing the sudden pressure drop acting on the model. The case with the back slant angle of 25° is calculated here as this is the angle at which all RANS models fail to correctly predict. Predicted streamlines with the ζ -f model are shown in Fig. 4.

The ζ -f model predicts the complete rear part in the separation region starting near the top edge. There is also no separation at the front part as shown in some latest measurements, see Lienhart et al. (2002), and Spohn and Gillieron (2002).

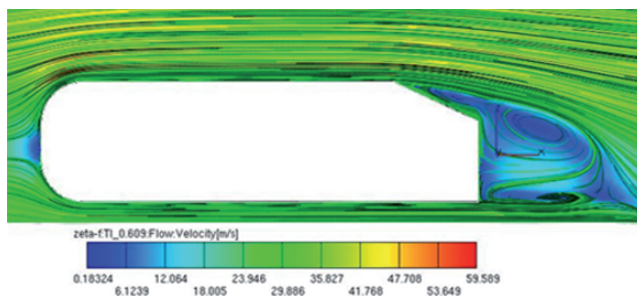


Fig. 4 Predicted streamlines by the ζ -f model

In the calculations performed with the PANS, the flow attaches to the slant again as reported by measurements. The PANS also captures the separation at the front part of the body as shown in Fig. 5. The selected velocity profiles for comparisons with the measurements are placed at the middle plane on the slant of the body at $x=-0.183$, -0.103 and -0.003 m respectively as sketched in Fig. 6. Both the streamwise and normal velocity components are not captured well with the ζ -f model at all chosen locations. On the other hand, the PANS predicted all profiles very close to the measured values as shown in Fig. 7. It is important to note that the same computational mesh consisted of 8 million cells (an unstructured mesh with hexahedral, tetrahedral and prismatic elements) was used for all calculations.

The same procedure was repeated for the BMW model, but this time using 4 million computational cells following work of Basara et al. (2007). In that work, the computational domain was meshed with two grids containing 2.85 million and 4 million hexahedral cells. Coarser grids, comprising 1.3 and 2.5 million, cells were also tested. The region around the car body was refined for the y^+ -values of the wall-nearest grid cells. This was achieved by combining blocks of structured meshes connected with arbitrary interfaces. The finest grid provided y^+ -values in the range of 10-30, while for other meshes y^+ -values were in the range of 20-60 and above (as shown in the work of Basara et al. 2007). The grid was optimized in order to allow future calculations including the truck model on relatively small mesh sizes. Fig. 8 shows the instantaneous vorticity as calculated by using PANS. This relatively coarse mesh, considering mesh requirements for the Large Eddy Simulation, was again fine enough that the dynamic parameter f_k depart from the RANS value 1, so the strong transient motion at the end of the model, was captured. The first comparisons with the measurements were made for the time averaged pressure coefficient C_p at the center plane of the model. At the mid part of

the model, the calculated pressure coefficient is over-predicted on the upper side, see Fig. 9. But in general, there is a good agreement with the measured data. Comparing the predicted and measured pressure coefficients on the side of the car, $Z=0.28$, as shown in Fig. 9, it is again clear that the main differences appear in the description of the transient wake behind the model. Nevertheless, this was the main reason to introduce PANS method in the calculations of the passing maneuvers where the transient effects might have an important role, see Fig. 10.

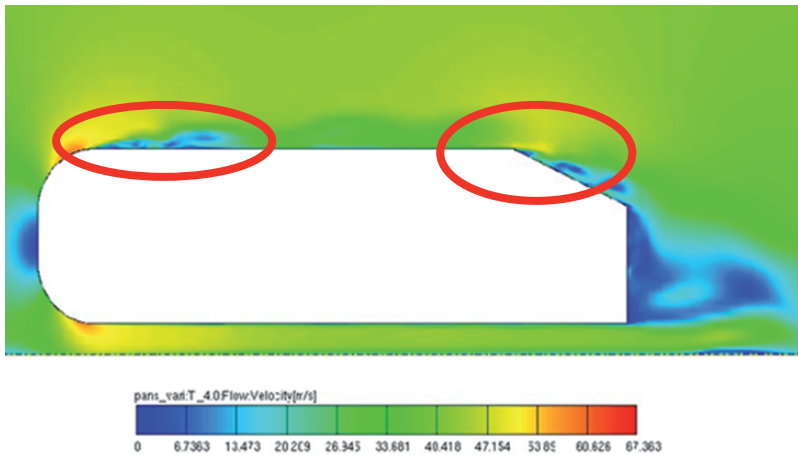


Fig. 5 Instantaneous velocity magnitude as predicted by PANS

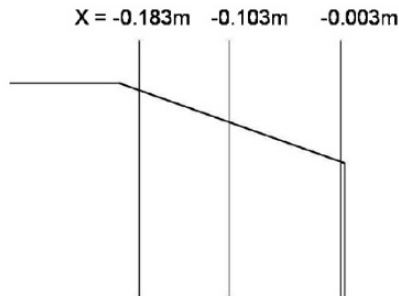


Fig. 6 Selected positions for the comparisons of velocity profiles

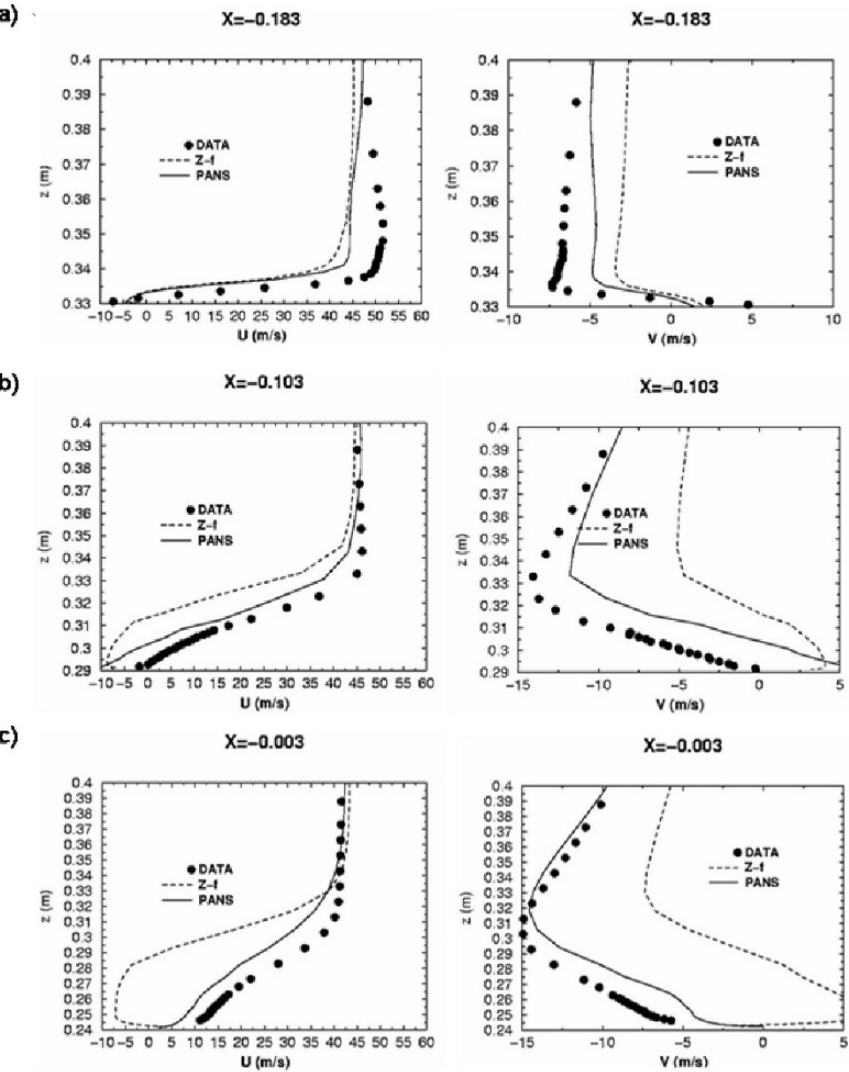


Fig. 7 Predicted mean and normal velocity components as predicted by the ζ -f model and the PANS: (a) -0.183m; (b) -0.103m; and (c) -0.003m

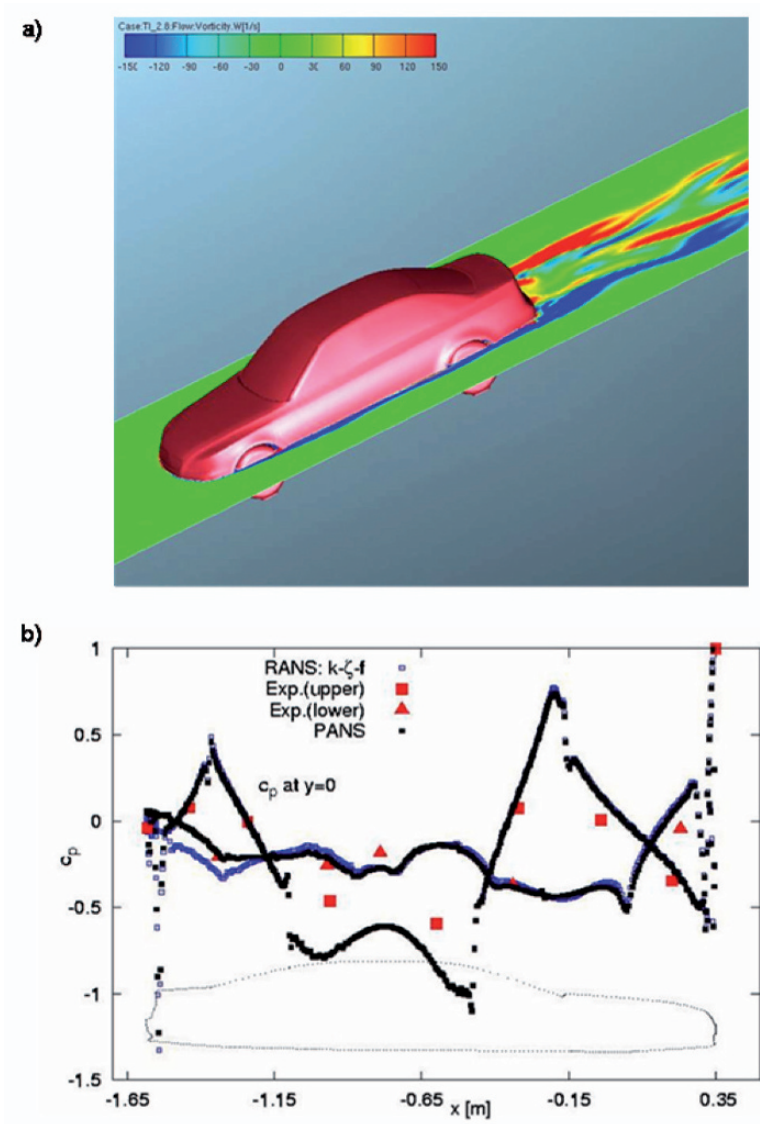


Fig. 8 Predicted an instantaneous vorticity (a) and a mean pressure coefficient at the center plane of the model (b)

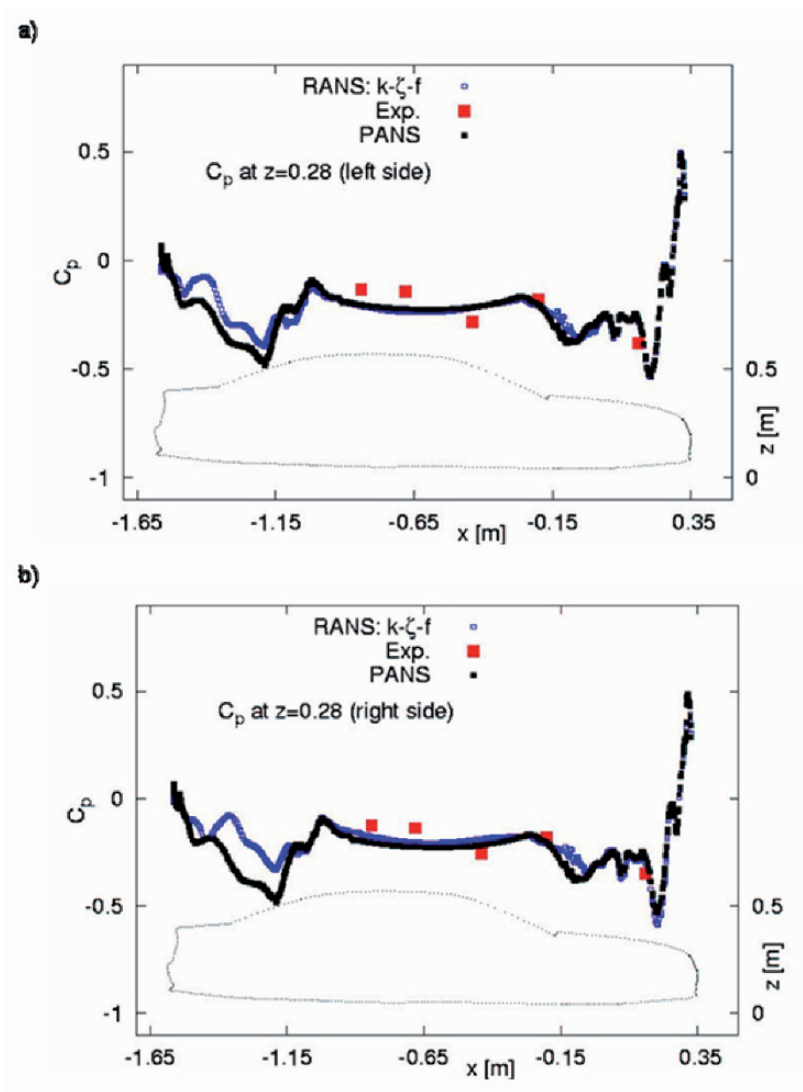


Fig. 9 Predicted pressure coefficient at the side plane $z=0.28$: (a) right side and (b) left side

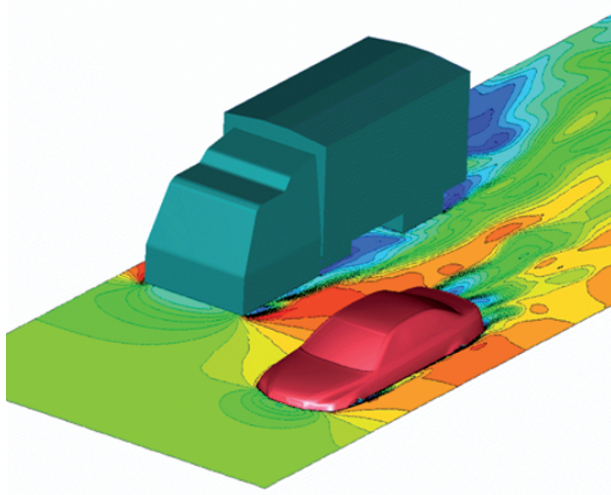


Fig. 10 A simulated passing maneuver showing contours of velocity magnitude

Conclusions

Potential of a recently proposed near-wall ζ - f turbulence model as well as the novel PANS approach for predicting the unsteady flow and associated turbulence around a 40%-scaled BMW car configuration including wheels were illustrated. PANS calculations were superior over the unsteady RANS calculations for the Ahmed Body, and therefore it is expected that this might bring an important difference for the simulation of transient passing maneuvers. A dynamic update of the PANS key parameter, the unresolved-to-total kinetic energy ratio, seems to be the promising approach to improve results on employed computational meshes. However, the both of the methods, exhibited good agreement with available experimental database for three-dimensional wall pressure mapping covering the car surface. The future work will include the simulation of the passing maneuvers with static meshes following presented measurements but also with moving meshes needed for the comparisons with the recent road measurements.

References

- AVL AST (2006) FIRE Manual 8.5. AVL List GmbH. Graz.
- Basara B (2006): An Eddy Viscosity Transport Model Based on Elliptic Relaxation Approach, AIAA J., Vol. 44, pp. 1686-1690.

- Basara B, Przulj V and Tibaut P (2001): On the calculation of external aerodynamics: Industrial Benchmarks., SAE 2001-01-0701.
- Basara B and Tibaut P (2002): Time dependent vs. steady state calculations of external aerodynamics, Proc. of the Aerodynamics of Heavy Vehicles: Trucks, Buses and Trains, Lecture Notes in Applied and Computational Mechanics, Vol. 19, pp. 107-119, Springer.
- Basara B, Aldudak F, Jakirlic S, Tropea C, Schrefl M, Mayer J and Hanjalic K (2007): Experimental investigations and computations of unsteady flow past a real car using a robust elliptic relaxation closure with a universal wall treatment", SAE 2007-01-0104.
- Baxendale A, Graysmith J, Howell J and Hayness T (1996): The CFD Investigation of Flow Separation over a Simple Vehicle Model, Proc, RaeS Vehicle Aerodynamics Conference, Loughborough University, U.K.
- Durbin P (1991): Near-Wall Turbulence Closure Modelling Without Damping Functions, Theoret. Comput. Fluid Dynamic., Vol. 3, pp. 1-13.
- Girimaji S, Srinivasan R and Jeong E (2003): PANS Turbulence Models For Seamless Transition Between RANS And LES: Fixed Point Analysis And Preliminary Results. Int. 4th ASME_JSME Joint Engineering Conference, Honolulu, Hawaii, FEDSM2003-45336.
- Girimaji S and Abdul-Hamid K (2005): Partially -Averaged Navier-Stokes Model For Turbulence: Implementation And Validation. AIAA Aerospace Sciences Meeting and Exhibit, Reno, Nevada, 2005-0502.
- Hanjalic K, Popovac M, Hadziabdic M (2004): A robust near-wall elliptic-relaxation eddy-viscosity turbulence model for, Int. J. of Heat and Fluid Flow, 25, 1047-1051.
- Khalighi B, Zhang S, Koromilas C, Balkanyi S, Bernal P, Iaccarino G and Moin P (2001): Experimental and computational study of unsteady wake flow behind a bluff body with a drag reduction device, SAE 2001-01-1042.
- Lakshmipathy S and Girimaji S (2006): Partially-Averaged Navier-Stokes Method for Turbulent Flows: k- ϵ model Implementation, AIAA Aerospace Sciences Meeting and Exhibit, Reno, Nevada, 2006-119.
- Lienhart H and Becker S (2003): Flow and Turbulent Structure in the Wake of a Simplified Car Model. SAE 2003-01-0656.
- Makowsky F and Kim S (2000): Advances in External-Aero Simulation of Ground Vehicles Using the Steady RANS Equations, SAE 2000-01-0484, Detroit.
- Popovac M, Hanjalic K (2005): Compound Wall Treatment for RANS Computation of Complex Turbulent Flows, Third MIT Conf. On Comp. Fluid and Solid Mech., edited by K.Bathe, Vol. 1, Elsevier, 802-806.
- Speziale C, Sarkar S, Gatski T (1991): Modeling the Pressure-Strain Correlation of Turbulence: An Invariant Dynamical Systems Approach, J. Fluid Mech., Vol. 227, pp. 245-272.
- Széchényi E (2004): The Overtaking Process of Vehicles, Progress in Vehicle Aerodynamics III-Unsteady Flow Effects.
- Spohn A and Gillieron P (2002): Flow separations generated by a simplified geometry of an automotive vehicle. IUTAM Symposium, Toulouse, France, 227, 245-272.
- Yamamoto S (1996): Aerodynamic Influence of a Passenger Vehicle on the Stability of the Other Vehicles, JSAE9730074.

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

Mark Page

Swift Engineering, USA
mpage@swiftengineering.com

Abstract Swift Engineering designs and builds high performance racecars and aircrafts. Techniques developed in this fast paced environment are relevant to transport industry. Experimental and analytical design example will be presented.

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

Jason Leuschen and Kevin R. Cooper

National Research Council, Ottawa, Canada
jason.leuschen@nrc.ca

Introduction

The National Research Council of Canada (NRC) has completed an evaluation of several prototype and commercially-available drag reduction devices for Class-8 tractor-trailer combinations. The evaluation focused on potential reductions in fuel consumption based on the results of full-scale wind-tunnel tests. The following is a summary of the results more fully presented in SAE 2006-01-3456.

Three primary devices have been evaluated, with the combination able to reduce fuel consumption by approximately 6,667 liters (1,761 US gal) annually, based on 130,000 km (81,000 miles) traveled per tractor at a speed of 100 km/hr (62 mi/hr).

Test program Summary

The test program was run in March 2006 in the 9-meter wind tunnel of the NRC, located in Ottawa, Canada. This is a closed-wall, closed-return, atmospheric wind tunnel having a test section that is 9.14 m high, 9.14 m wide and 22.5 m long. It has a maximum speed of 200 km/h (120 mi/h) and a turbulence level of 0.5 percent.

The tractor drive wheels sat on pads attached to the main balance below the tunnel floor. The front tractor wheels and the trailer bogey were floated on low-profile air bearings to allow the balance to measure the wind-axis drag and side forces, and to permit the turntable to rotate up to a yaw angle of at least 13°. Both forces are necessary to compute the body-axis drag.

The installation used in this test is seen in Fig. 1. The tractor under test was a Volvo VN 660 provided by Robert Transport Inc. of Quebec, Canada. The trailer could be configured to be either 8.5 m (28 ft) or 12.2 m (40 ft) long by removing a section of the body. The kingpin was set for a 1.14 m (45 inch) gap between the front face of the trailer and the rear face of the cab. The side corners at the front of the trailer are rounded with a 127 mm (5 inch) radius but the top edge is square.



Fig. 1 Volvo VN 660 and 28-ft. Trailer in the NRC 9m x 9m wind tunnel

Discussion of results

The data presented in this paper are given as the wind-averaged drag coefficient at 29.6 m/s (107 km/h, 65 mi/h). The procedure for the calculation of the wind averaged drag coefficient comes from [6] and is summarized in [1].

Based on the change in wind-averaged drag, the expected fuel-savings is calculated from,

$$\Delta\mu(V_t) = 4.25 \times 10^{-4} V_t^2 \Delta \overline{C_D}(V_t) A \text{ L/100 km} \quad (1)$$

Where A was 10.9 m^2 (the value used in calculating the drag coefficients) and V_t was taken as 100 km/h (62 MPH). The constant in equation (1) is based on a transmission efficiency of 0.85, an air density of 1.225 kg/m^3 and an average

specific fuel consumption of 0.275 liters/kW-h (0.042 US gal/hp-h) and a unit conversion factor of 1.072.

It is possible to calculate the annual fuel savings realized by installing these aerodynamic devices for an average fleet. Average annual distances covered by heavy trucks are now in the range of 200,000 km (125,000 mi), but only a portion of this distance is covered at cruise speed. For the purposes of this paper, the distance covered at cruising speed will be assumed to be 130,000 km (81,000 mi). This will result in a conservative estimate of fuel-savings because these devices continue to provide fuel savings at lower speeds, which has been neglected here. Thus, the annual fuel saving is 1,300 times that for the 100 km distance given in equation (1).

The standard vehicle components that were evaluated are shown in Figs. 2 and 3. The drag and resulting annual fuel consumption changes are summarized in Table 1.

Table 1 Tractor Component Drag and Fuel Increments

	$\overline{\Delta C_D}(100 \text{ kph})$	Annual fuel savings (L)
OEM side mirrors	-0.016	-938
OEM bug deflector	-0.015	-903
OEM fender mirrors	-0.010	-588
engine cooling inlets blocked	0.000	0
sun visor w/ roof deflector	0.001	54
hub caps (truck & trailer)	0.002	120
deer bumper	0.002	120
wrap-around splash guards	0.005	292
prototype roof deflector filler	0.014	825
fifth wheel forward 254 mm	0.016	982
OEM tank and cab skirts	0.027	1,596
OEM side extenders	0.042	2,499
OEM roof deflector	0.072	4,318



Fig. 2 Common components whose drag effects were measured



Fig. 3 Common components whose drag effects were measured

It can be seen that many components produce noteworthy drag and fuel changes. In particular, the bug deflector raises fuel consumption and the mirrors are also an area worth study. Mirrors have been singled out since they are a large source of drag that are dictated by current safety regulations. It would be possible to eliminate mirror drag if regulations were changed to allow video cameras to replace mirrors as the means of providing rearward vision. The price of the video system would be covered by fuel savings, and the cameras may increase safety by offering a larger field of view with infra-red capabilities that can penetrate fog, rain or darkness.

All of the other components are beneficial, especially the standard cab-roof fairing, cab side extenders and tank skirts, which form the current aerodynamic package. The sun visor was an OEM product, its design obviously having been developed in conjunction with the roof deflector, so it actually produced a small drag reduction. This may not be the case if the sun visor is used with trucks without a roof deflector.

Eliminating the cooling flow, by covering the front grille and the lower intake in the front bumper, had a negligible effect on drag. This has been observed in several full- and model-scale tests of modern tractors.

Prototype hub caps consisting of solid metal disks on the outside wheels of the trailer and tractor showed a negligible change in drag. Recall that this test was performed with fixed wheels and the effects of wheel rotation on the results are unknown. Furthermore, these disks could be expected to have a harmful effect on brake cooling.

Another prototype considered was a panel to fill the large cut-out on top of the roof deflector that provides space for the trailer during sharp cornering maneuvers. The fuel savings from this panel are significant and it is envisioned that a simple, lightweight, flexible panel should be able to provide this function at low cost.

Finally, the effect of reducing the tractor-trailer gap was studied. There are operational issues to be addressed in doing this, such as axle weights, ride quality and turn clearance, but the potential savings are significant for no capital cost. The expected change in drag depends upon the original gap size. In this case, the gap was reduced from 1.14m (45 in) to 0.89m (35 in). If the original gap were larger or smaller than 1.14m, the drag reduction may be smaller or larger, respectively, than that measured here. A related note is that the optimum angle of the tractor roof deflector is partly a function of the gap size. Thus, a significant change in gap size should be accompanied by a re-optimization of the roof deflector. In previous studies an overall drag increase has been observed after significantly reducing the gap without adjusting the roof deflector, likely because the airflow is under-deflected and strikes the trailer which is now closer.

The results presented in Table 1 are application specific and will only apply strictly to the particular model of tractor and component tested. Nevertheless, the results are useful as guidelines to the expected magnitudes in similar applications. As an example, the deer bumper showed a slight reduction in drag, a surprising result considering its large form. This shows that a properly designed and positioned deer bumper can provide a drag reduction.

The new components that were evaluated are shown in Figs. 4 and 5 and aerodynamic drag and fuel results are summarized in Table 2.

Table 2 Performance of new add-on components

	$\overline{\Delta C_D}(100 \text{ kph})$	Annual fuel savings (L)
<u>base drag reduction</u>		
Transtex Composite folding rear trailer deflector	0.0506	3,047
Aerovolution inflatable rear trailer fairing	0.0438	2,638
Trailer vortex strakes	-0.0195	-1,174
<u>trailer leading edge fairings</u>		
Freight Wing NXT Leading Edge Fairing wo/ roof deflector	0.0369	2,222
Manac prototype trailer leading edge fairing	0.0335	2,015
Freight Wing NXT Leading Edge Fairing w/ roof deflector	-0.0019	-114
<u>underbody drag reduction</u>		
Freight Wing Belly Fairing (low rider)*	0.0478	2,879
Laydon Composites main and rear skirts*	0.0391	2,355
Laydon Composites main skirts*	0.0376	2,264
Freight Wing Belly Fairing*	0.0367	2,210
Francis Cardolle trailer bogey fairing	0.0145	872
Francis Cardolle trailer wheel fairings	0.0078	470
<u>gap sealing</u>		
Laydon Composite trailer nose fairing	0.0135	813
Volvo cab side extender extensions	0.0123	741
Labyrinthine truck-trailer gap seal	0.0018	110

* Modified to fit a 40-foot trailer.

The large region of separated flow at the rear of a van-style trailer is the largest untreated source of drag on a modern tractor trailer. Not surprisingly, this is the area where the greatest reductions in drag were found. Two devices were tested in this area, the inflatable Aerovolution and triple-panel Transtex boattails. Each device addresses the issues of access to the trailer doors, though in different manners, while producing significant reductions in drag. These devices would be complementary to trailer skirts, and vice versa, and the choice between them comes down to non-aerodynamic issues.

The vortex generators at the rear of the trailer increased overall drag. The measured base pressure did not change, so the extra drag must be tare drag on the angle sections. They were made from 51mm x 51mm aluminum angle, 914 mm long, installed six per side on the trailer, 30° nose up from the horizontal. Four were mounted on the roof in an asymmetrical arrangement about the trailer centerline.

As has been seen before [2], a fairing on the front top face of the trailer provides little additional benefit when a cab-roof deflector is present. However, if a

roof deflector is not present, a leading-edge fairing can provide significant fuel savings, although only about half as much as a full roof deflector. This may be a consideration for trucks that often operate without a van trailer, for instance switching between flatbeds and vans, when a fixed roof deflector may increase the drag with a flatbed.

Both brands of trailer skirts tested showed similar drag reductions to each other and to those previously reported [1, 2]. The rear skirts, behind the trailer bogeys, showed no reduction in drag, although they may be effective when paired with a boat-tail [2]. It should be noted that the skirts were not installed as prescribed by the manufacturers. Ground clearance was approximately 100 mm (4 in) greater than normal due to the air bearings, which would decrease the drag reduction. The trailer was 12.2 m (40 ft) in length instead of the typical lengths of greater than 14.6 m (48 ft) that the skirts are designed for.

A lower skirt provides better performance, as shown in [1]. Previously [1,2], skirts had been tested on an 8.5 m long trailer with results close to those measured with the 12.2 m trailer presented here. Thus, it is expected that trailer length is not a critical parameter since most of the drag reduction comes from sheltering the trailer bogey.

A fairing on the trailer bogey was less effective than the skirts. This suggests that the skirts may also reduce the drag on the rough underbody of the trailer. It would be interesting to cover the trailer floor ribs to verify this.

The labyrinthine gap seal provided almost no drag reduction. The seal was made up of six 305 mm deep 17 mm thick plywood panels that were 2.44 m (8 ft) high. It would appear that they did not significantly block the gap flow, and only served to move the trailer front face closer to the tractor. The Laydon Composites Nose fairing provides a modest drag reduction and would be useful in situations where complete gap closure is not possible. Fully closing the gap is clearly the target to aim for, although partial closure through additional cab-extender extensions is simple and still beneficial.



Transtex Composites folding rear trailer deflector



Vortex generators (side)



Freightwing NXT leading edge fairing



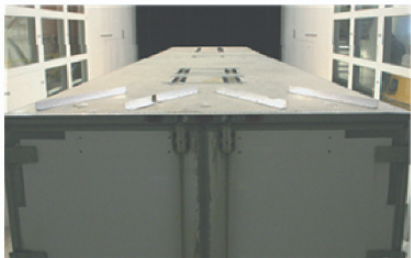
Freightwing belly fairing (low rider extension shown unattached)



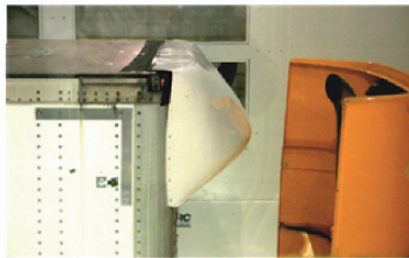
Laydon Composites skirts



Aerovolution inflatable boattail



Vortex generators (top)



Menac prototype leading edge fairing

Fig. 4 New add-on components



Fig. 5 New add-on components

The results obtained for the production devices were sensibly like those found in the development tests. The combination of: tractor-mounted gap sealing with the largest available side extenders, trailer side skirts and trailer boat-tailing combine to provide a total drag reduction of $\Delta C_D(107) \approx 0.111$. Payback periods will vary based on fleet statistics, but the savings for any particular operation mode can be calculated with the data in Tables 1 and 2 using Equation (1). The critical parameters are fuel cost, device cost, annual cruise mileage and cruise

speed. Keep in mind that while fuel savings will increase with speed, overall consumption will increase by a greater amount.

Conclusions

Full-scale wind tunnel measurements were made on new production add-on devices meant to reduce the aerodynamic drag of Class-8 tractor-trailers beyond that provided by the current aerodynamic package. They consisted of devices to reduce the tractor-trailer-gap drag, to reduce trailer underbody and bogey drag, and to reduce trailer base drag.

Simple estimates of fuel consumption at steady speed on a flat road indicated annual fuel savings of 6,667 liters (1,761 US gallons). This estimate was predicated on a cruising speed of 100 km/h (62 mi/h) and 130,000km (81,000 miles) traveled annually at this speed.

Acknowledgments

The National Research Council of Canada greatly appreciates the financial support of Natural Resources Canada in this study. It also appreciates the support of Transport Robert Inc and the suppliers of the new hardware. Finally, The NRC thanks its colleagues involved in the US DOE-sponsored aerodynamic drag reduction programs for the moral and intellectual support that they have generously provided.

References

1. Cooper K – Truck Aerodynamics Reborn – Lessons from the Past. SAE Truck and Bus Conference and Exposition, SAE 2003-01-3376, Houston, TX, Nov. 2003.
2. Cooper K & Leuschen J – Model and Full-Scale Wind Tunnel Tests of Second-Generation Aerodynamic Fuel Saving Devices for Tractor-Trailers. SAE 2005-01-3512. SAE Commercial Vehicle Engineering Conference, Chicago, IL, USA, Oct. 2005.

3. Cooper K (ed.) – Wind Tunnel Wall Corrections for Automotive Shapes in Closed-Jet Wind Tunnels. Society of Automotive Engineers Special Publication SAE SP-1176, Warrendale, MI, USA, Feb. 1996.
4. Leuschen J & Cooper K - Full-Scale Wind Tunnel Tests of Production and Prototype, Second-Generation Aerodynamic Drag-Reducing Devices for Tractor-Trailers. SAE 2006-01-3456.
5. Wood R & Bauer S – Simple and Low-Cost Aerodynamic Drag Reduction Devices for Tractor-Trailer Trucks. SAE 2003-01-3377, SAE Truck and Bus Conference and Exposition, Houston, TX, USA, Nov. 2003.
6. Cooper K (ed.) – SAE Wind Tunnel Test Procedure for Trucks and Buses. SAE Recommended Practice J1252, Aug. 1979.

A Fleet Operator's Perspective on Commercial Vehicle Drag Reduction

Réjean Laflamme

Robert Transport, Canada
rlaflamme@robert.ca

Abstract The research establishment has proven the potential fuel-savings for many devices/modifications for commercial trucks, however many of these developments remain in limited use. The primary reason that these substantial fuel-savings are not being realized is that fleet operators have chosen not to adopt them in their current form. Why have roof fairings been accepted while trailer skirts haven't? The reasoning behind the selection is logical, and often not obvious, so to succeed the development community must understand the environment of the fleet operator and respond with appropriate technology. This paper will address the considerations of a typical fleet operator when evaluating new technologies. A discussion and examples of the following issues will be given:

1. Introduction to Robert Transport: size of fleet, typical equipment, types of cargo, description of routes.
2. Operating Costs:
 - a. Cost of installation, maintenance and operation in terms of cash and lost productivity.
 - b. Weight of the device vs. cargo when the amount of paying cargo is limited by weight not volume.
 - c. How do we measure the savings from the device? Track tests are expensive and relatively inaccurate and fleet tests can be statistically irrelevant.
3. Operating Procedures: How compatible are the devices with current roads, shipping yards and loading facilities?
4. Human Resources: There is a shortage of qualified drivers so it can be difficult to retain drivers if the technology adds to the driver's workload, is unpleasant or adversely affects driver comfort.
5. Reliability: The cost of unplanned downtime is significant, so the technology is expected to operate reliably, for 10 years or longer, in a variety of harsh conditions including extreme temperatures, wind, road salt, dust. Another consideration is resilience to damage caused by the drivers.
6. Safety: Visibility and crash worthiness.

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

Kenneth D. Visser, Kevin Grover

Clarkson University, USA
visser@clarkson.edu

Abstract Over-the-road tests have been conducted to evaluate and optimize sealed single and dual open aft cavity geometries for drag reduction on Class 8 tractor-trailers. Several tractor-trailer combinations and methods were used over various routes in northern New York. Additional testing on the most promising geometries was carried out by Volvo Technology of America, Inc. at the Transportation Research Center Inc. in Ohio according to the Recommended Practice J1321 Joint TMC/SAE Fuel Consumption Test Procedure - Type II. An over-the-road improvement of 0.63 mpg was observed in the back to back runs in New York, which corresponds to an estimated fuel savings of about 10% for present day class 8 loaded tractor-trailers. Reported results from the SAE test indicated a fuel savings of 3.9% to 4.7% which is estimated to be approximately 0.46 to 0.57 mpg based on the as tested empty tractor-trailer configurations. The optimal geometry for the sealed aft cavity was found to be 15 deg inset on the side top panels, and a 7 deg inset on the bottom. It was observed during the New York tests that a noticeable shift in the data occurred over time, attributed to the warming of the fuel during the test, and this underscores the necessity for back to back run testing.

A Novel Approach to Heavy Vehicle Drag Reduction

Alec Wong¹, Kevin Horrigan²

¹ Kenworth Truck Company, USA

² EXA Corporation, USA

alec.wong@paccar.com

Abstract The recent volatility of fuel prices and the potential of explicit CO₂ emission control regulations have caused aerodynamic performance to become a critical component of heavy vehicle development. The use of traditional computational fluid dynamics (CFD) simulation for aerodynamic drag prediction on fully complex production geometries has not been adopted for a number of key reasons: the high level of expertise required, simulation time-to-completion, and accuracy versus experiment. This paper demonstrates that the proprietary, Lattice-Boltzmann based simulation method of EXA Corporation delivers valuable results on fully complex geometry, allowing aerodynamic optimization in the concept and design phase. Included is an overview of the simulation method and results for two fully detailed open grille Kenworth models: the T603 and T2000. Comparisons are made to full scale drag meter testing. Additional results are provided for several trailer drag reduction devices including a full gap seal, side skirts, boat tail, and wake boards. Advanced post-processing of the simulation results is used to study transient flow structures and highlight the physics involved with the drag reduction of these devices.

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

Linus Hjelm and Björn Bergqvist

Volvo 3P, Sweden
linus.hjelm@volvo.com

Abstract The aerodynamic situation for trucks on the European market differs from that in North America on a number of points. Perhaps the most significant difference is that in Europe trucks are of the CoE configuration (Cab over Engine) and in North America trucks are of the conventional type with a hood. Another major difference is that trucks in Europe are speed limited to 90 km/h (56 mph) which of course means that aerodynamics as a whole has less of an impact there. These differences are primarily dictated by different legislations, which in turn have a lot of different side effects. This paper will high-light some of the differences and their impact on aerodynamics, as well as taking a look at possible future ideas such as: extended front or short nose, ride height adjustments, convoy driving, etc.

1 Introduction

Historically most trucks have been of the conventional type. But with the increasing need to make transportation more effective, maximizing load at the expense of space for the driver, has predominantly in Europe, resulted in trucks with the Cab placed on top of the engine (CoE). All these constraints are regulated by legislations. These legislations are, and have been, very different in European compared to in North America. (There are differences in other parts of the world but those will not be considered here).

Of all different legislations it is the ones concerning the length, height, width, weight which has had the biggest impact on the over all shape of the trucks, and hence the aerodynamics. But there are also differences in culture, infrastructure and geography which have contributed in making the detailed design of trucks on the two continents different.

2 Length

The regulation which has most impact on the over all design is the one regarding the length of the vehicle. In Europe all lengths, depending on truck-trailer combination and depending on country, are measured from the most forward point to the most rearward point of the whole vehicle. In North America length regulation does not restrict the total over all length in the same way (see Fig. 1). This is the reason why, in Europe the cabs are of the more compact so called CoE type and in North America they are of the conventional type with a hood. A conventional truck is aerodynamically better than a CoE. For example the difference between a CoE Volvo and a conventional Volvo is of the order of around 0.05 in C_D .

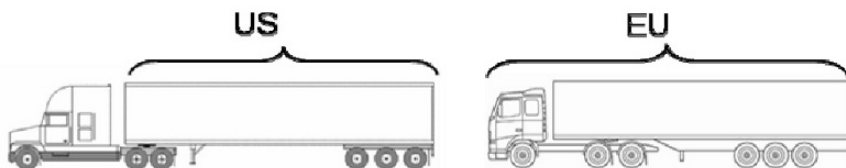


Fig. 1 Different ways of measuring length

When it comes to the cab shape itself this constraint has meant that for a truck OEM in Europe you are left with a fairly limited number of parameters you can work with, for example: Windscreen rake, corner radii, wedge angle, roof shape, underside. And the freedom to work with these parameters is very limited. On one hand you have the length and width regulations and on the other hand you have a constantly increasing need for interior volume and packaging space for all components inside.

Normally there is little extra room to work with on the lower part of the front when it comes to shape, as it is very packed with cooling unit, head lamps, washer liquid bottles etc. All European trucks are also fitted with so called dirt deflectors on the lower corners. If these dirt deflectors are aerodynamically beneficial or not depends on both the design of the detail itself as well as on the design of the cab in this area.

The main priority is to ensure enough corner radius to make the flow stay attached. The part on the lower part of the truck which allows for the biggest room for modifications is probably the bumper/spoiler/underside.

The area where there is still a little bit more freedom to work with is the roof shape and windscreen angle, in other words, the top of the cab. The trend however is that the cab sizes, the height of the cabs, are getting higher and higher, due to increasing need for interior volume. Even here there will be less and less to work with.

As conventional trucks are less restricted in lengths there is more room for design changes on them compared to a CoE truck.

Another indirect effect of the different type of length legislations is the size of the gap between the back wall of the cab and the front face of the semitrailer. Due to the over all length regulation this is kept to a minimum in Europe to ensure as much loading capacity as possible, while still maintaining full mobility between cab and trailer. In North America these gaps are usually much bigger, sometimes up to three/four times bigger. And not surprisingly big gaps have a major negative impact on the C_D -value (see Fig. 2).

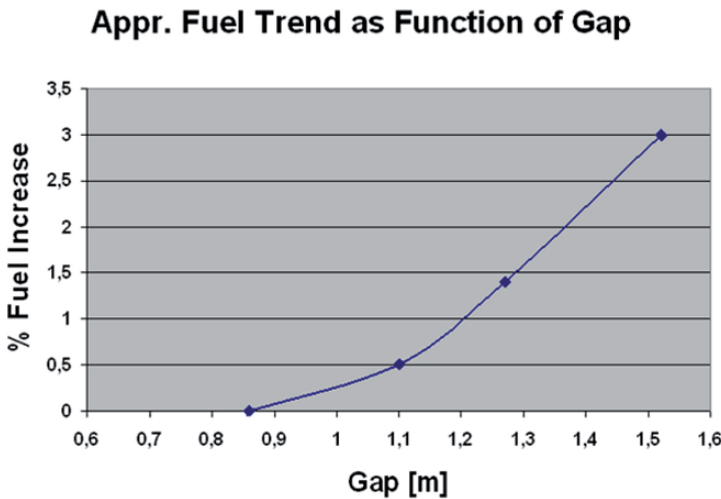


Fig. 2 Importance of gap for fuel economy. Tested on a Volvo VN $\frac{1}{2}$ -scale model truck + trailer during tests in the Volvo wind tunnel (PVT) in Gothenburg

3 Height

In Europe there are a lot more variations in the total height regulation, from country to country than in North America, and it can differ with as much as 1 m. This has led to that adjustable roof fairings are more or less standard on all European trucks. An adjustable roof fairing requires roof fairing setting instructions. In Fig. 3 one example of such a setting instruction can be seen. The user measure the distance between the rear wall of the cab and the front wall of the trailer (G), and the vertical distance between the top of the trailer and the top of the cab ($H=H1-H2$). These numbers are then put in the table, and the appropriate fixing in the adjustment details can be found.

Globetrotter FH

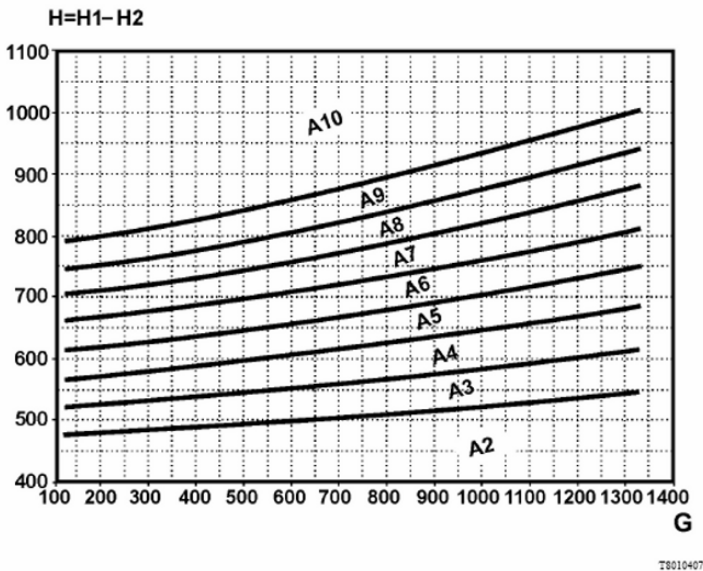
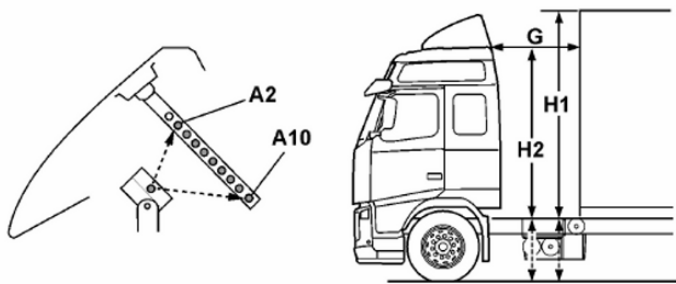


Fig. 3 Roof fairing setting instruction. Example from the Driver Handbook [1]

4 Width

There is one standard over all width in North America, 2.59 m. In Europe there are three; 2.50, 2.55 and 2.60 m depending on country and on type of transport. In

Europe the width of the cabs are made as close to 2.50 m wide as possible in order to maximize the interior volume. In North America the cabs are usually made a bit narrower to ensure easy ingress and egress to the cab.

5 Speed

One regulation which has a big impact on aerodynamics, but perhaps not so much on the actual design of the trucks is the speed. In Europe trucks are electronically limited to 90 km/h (56 mph), whereas in North America considerably higher speeds are common. During our tests the drag becomes Reynolds number independent at just over $4 \cdot 10^6$, equivalent to a speed around 70 km/h (see Fig. 4).

The higher speeds in North America means that aerodynamics account for a larger part on the over all fuel consumption and the over all cost for the truck owners.

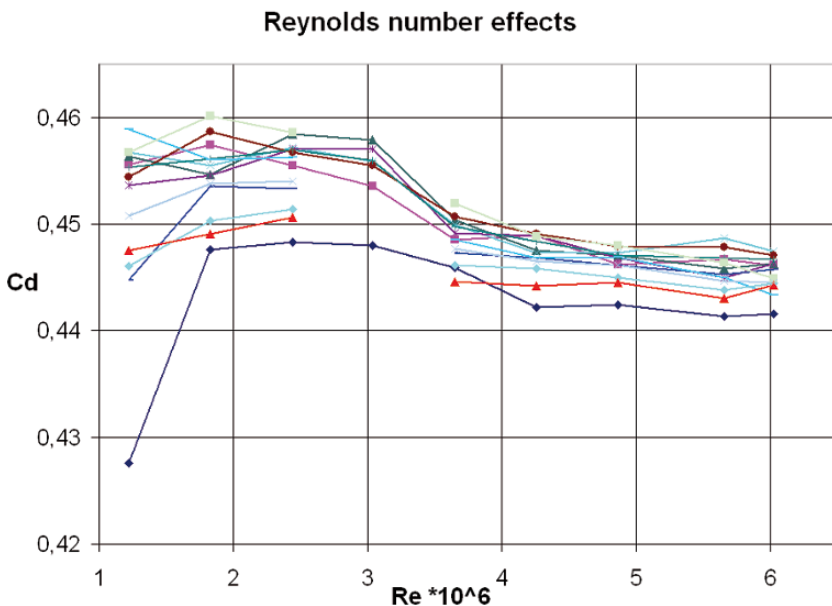


Fig. 4 Reynolds number effects. Each line corresponds to a different 1/2-scale model truck configuration tested at different speeds in order to investigate the Reynolds number effects. Reference length is the square root of the projected frontal area

6 Trailer

It's not only the tractors that are different, so are the trailers. The vertical corner radii are well rounded in North America while they have sharp or angled corners in Europe. But because the gap is smaller in Europe the side fairings on the cabs can be fitted fairly well with the trailer so they somewhat reduce the bad effect of having sharp corners.

The underside of the trailers are usually quite different, in North America the undersides are usually flat and “empty” whereas in Europe they have beams and are also usually fitted with tool boxes, spare wheels etc. And for safety reasons trucks in Europe have to have under run protections around all sides of the vehicle, even on the trailers. These things help to reduce the C_D because they all act a bit like chassis fairings on the trailer.

7 Future

The C_D for a CoE truck-semitrailer combination today is somewhere around 0.6 – 0.65. Unfortunately this number will not decrease much unless the legislation and trucking/truck transport business change. The aerodynamic evolution is getting closer and closer to the flat part of the development curve when it comes to basic cab shape for CoE – trailer combination. In Fig. 5 the aerodynamic development of a number of different CoE Volvo truck models from the seventies up until today can be seen.

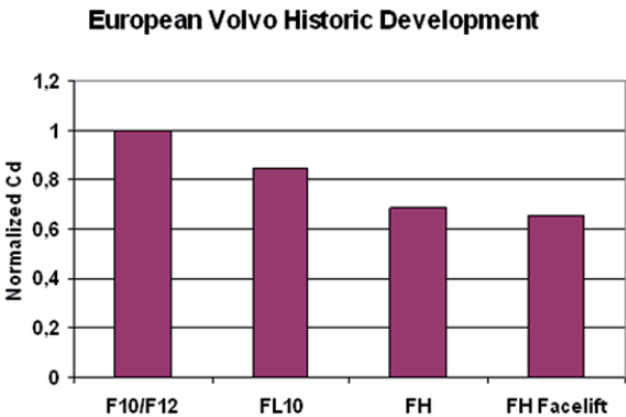


Fig. 5 Aerodynamic development of CoE Volvo trucks. F10/F12 is a model from the seventies and the FH is the current model

Once a good cab shape is achieved, and roof fairings, side fairings and chassis fairings are fitted, the easy gains have been achieved.

In order to reach significant improvements the trailers have to be worked with, or ultimately, treat a tractor trailer combination as one unit.

However there are still some potential areas on the cab and perhaps the most interesting one is the so called; Crash zone, or Soft nose, on CoE (see Fig. 6). Due to safety reasons, discussions have been going on in Europe about introducing an extension of the front face below the wind shield of around 300 mm. This should act as a deformation zone to protect or to minimize injuries to pedestrians, bicyclists and other traffic. Such a “nose” could of course provide a chance to design the front of the truck more aerodynamically efficient. Some initial tests have shown that there seems to be a non negligible potential. The main problem for introducing this seems to have more to do with administration than with anything technical. To define the technical specifications for a deformation zone, and to avoid that this extra volume is not used to package “hard” things is the problem.



Fig. 6 Examples of “Crash nose” or “Soft nose”. Front face of the truck below the windshield extended approximately 300 mm forward.

The mirrors are another potential area for improvements. It is clear that it would be beneficial if they could be replaced by cameras. However, in some cases where flow separations occur at the A-pillar, the mirrors work as guiding vanes, directing the flow around the A-pillar so that the net drag of the truck actually decreases when the mirrors are added. In order to get a good benefit from replacing the mirrors with cameras, the cab has to be designed in such a way that separations are avoided at the A-pillars.

In North America the length legislations has recently changed to allow for another 2 or 4 feet of extra length at the rear of the trailer in order to make it possible to put add-on devices there. This extra length is to be used for fuel saving reasons only, not to store extra cargo. This of course opens a big potential for reducing the C_D . But the problem is that such a device easily gets in conflict with the rear door openings. Therefore a device like that needs to be made practical for the user, if it is ever going to be used.

Another feature on trucks, which have a big impact on aerodynamics, are the gaps between the bodies of tractor-semitrailer combinations and rigid-trailer combinations, which are necessary for the vehicle to be able to maneuver.

In fact, the usages for trucks are very different between when they are driving slowly/standing still, and when they are driving fast. It could be said that they have two modes; One when driving slow/standing still (when all the flexibility to maneuver in confined spaces is needed, during loading/unloading, and often good ground clearance is needed for curbs or when driving on and of ferries). The second mode is when driving at high speed (on a road when only a very small maneuverability is needed, and if the road is even and smooth very little ground clearance is needed). One idea could therefore be to make use of this and in some ways alter the shape/configuration between these two modes at a predefined speed for example.

One example is to have a coupling which can be shortened once you are on a motorway to decrease the gap between the different bodies in order to reduce the drag contribution from such gaps (see Fig. 7).



Fig. 7 Adjustable length of coupling

Another example could be the ride height of the vehicle; it could be lowered once you have reached a certain speed (see Fig. 8). You do not drive very fast on a bad or uneven road. This requires that the truck is equipped with air suspension on all axles, which only a limited number of trucks have today.



Fig. 8 Adjustable ride height

Inflatable gap-fillings between the rear wall of the cab and front face of the trailer is another idea. Iveco has shown that, as well as an inflatable boat tailing on one of their concept trucks [2, 3].

More and more trucks today are equipped with ACC (Adaptive Cruise Control), which makes it technically possible to drive very close behind another truck in order to make use of the aerodynamic benefits of moving in a close convoy. But to make that a reality the safety aspects have to be solved.

8 Conclusions

No doubt there is a huge potential to reduce the C_D on trucks. That has been shown by countless tests and simulations as well as by a number of concept trucks over the years. But in order to make a reality out of that, the trailers have to be worked with in one way or another.

That the trailers stand for a majority of the drag of a truck on the road is common knowledge. During the years a lot of resources have been spent on finding devices, shapes and solutions which will work aerodynamically. So the main problem is not how to make a truck more aerodynamically efficient, because we have good knowledge of that today, the real challenge lies in finding practical, robust, economic, low maintenance and not to forget, aesthetic solutions which will work in real life situations.

In order for that to be a reality, truck OEM:s, trailer OEM:s as well as the end customers need to find ways to cooperate more closely.

Acknowledgments

Michael Sorrells, Volvo 3P

References

1. "Driver Handbook", Volvo Trucks Corporation, 2007.
2. <http://195.200.115.136/Textbase/work/2007/vehicle/Durelli.pdf>
3. <http://www.roadtransport.com/Articles/2008/01/31/129674/15-better-mpg-with-transport-concept.html>

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

Mike Camosy, Andre Brown, Henri Kowalczyk, Gaylord Couthier

Auto Research Center, USA
mcamosy@arcindy.com

Abstract Extensive knowledge exists in the testing of 1/8th scale, Class 8 tractor trailer models within fixed floor wind tunnels. This size of model has proven to correlate well with in a Reynolds number ranging between 1M and 6M. Prior testing is well documented on a Generic Conventional Model (GCM) in the NASA AMES 7x10ft and 12ft wind tunnels. Within this prior research into Heavy Vehicle Aerodynamics, it has been identified on numerous occasions that improved accuracy will allow for greater understanding of the flow field structure in the underbody and internal flow regions. The aim of this paper is to summarize the new research conducted at the Auto Research Center (ARC), a 50% rolling road scale wind tunnel facility in Indianapolis, Indiana. ARC has tested a rolling road 1/8th scale tractor trailer at 50m/s constant dynamic (approximately 1M Re). The model features fully articulated suspension & axle systems, engine bay detail for internal flow simulation, realistic radiator blockage, and an automated model motion system capable of controlling the model in roll, heave, pitch & yaw. To remain consistent with the previous testing conducted by NASA AMES, a replica of the GCM tractor and trailer shapes were used for the upper aerodynamic surfaces. The test objectives initially included a demonstration of correlation between the ARC and NASA wind tunnels with the rolling road switched off. This baseline is then compared to data obtained with the rolling road on, allowing the impact of 18 rotating wheels & tires on the flow structure to be visualized and analyzed. Further studies are presented on the effects of improved detail in the underbody regions, and on the effects of internal & cooling flows as applied to the GCM model. Future work at ARC will use detailed cab bodywork for analysis of specific OEM tractors & trailers.

Author Index

<u>Author</u>	<u>Page Nos.</u>
Alajbegovic, A.	215
Aldudak, F.	433 – 446
Almbauer, R.	419 – 432
Arwatz, G.	115 – 133
Basara, B.	433 – 446
Bayraktar, I.	195
Bearman, P.	3 – 13
Bergqvist, B.	469 – 477
Bhatnagar, B.	213
Bodony, D.	19
Brown, A.	479
Camosy, M.	479
Castellucci, P.	383
Cattafesta, L.	151 – 160
Chang, J.	349 – 361
Chen, K. H.	21 – 38, 39 – 51, 53 – 74, 75 – 99
Cooper, K. R.	451 – 462
Corke, T.	135 – 149
Couthier, G.	479
David, S.	115 – 133
Dayan, I.	115 – 133
Dillmann, A.	233
Fono, I.	115 – 133
Ghuge, H. A.	363 – 381
Girimaji, S. S.	433 – 446
Greenblatt, D.	179 – 191
Grover, K.	465
Haff, J.	179 – 191
Ham, F.	19
Hewit, R.	135 – 149
Hjelm, L.	469 – 477
Horrigán, K.	215, 467
Hyams, D.	325 – 338
Iaccarino, G.	19, 21 – 38
Iser, F.	419 – 432
Jarkirlic, S.	433 – 446
Jinda, S.	21 – 38

Johnson, J. P.	21 – 38, 39 – 51, 53 – 74, 75 – 99
Johnson, R. W.	259 – 297
Khalighi, B.	19, 21 – 38, 39 – 51
Khalighi, Y.	19
Kowalczyk, H.	479
Krajnović, S.	197 – 211
Lafferty, C.	215
Laflamme, R.	463
Lee, R.	39 – 51
Leong, C. H.	53 – 74, 75 – 99
Leuschen, J.	451 – 462
Li, R.	239, 243 – 258
Loefeldahl, L.	179 – 191
Loose, S.	235
Menter, F.	323
Merati, P.	53 – 74, 75 – 99
Mitchell, B.	325 – 338, 339 – 348
Mittal, R.	151 – 160
Moin, P.	19, 321
Nayeri, C. N.	179 – 191
Nichols, S.	325 – 338
Orellano, A.	219
Ortega, J.	161 – 178, 317
Othman, H.	135 – 149
Page, M.	449
Pagliarella, R.	101
Palei, V.	115 – 133
Pankajakshan, R.	325 – 338
Paschereit, C. O.	179 – 191
Paul, J. C.	259 – 297
Plocher, D.	403 – 417
Pointer, D.	349 – 361
Radovich, C.	403 – 417
Roy, C. J.	363 – 381
Rueter, A.	237
Salari, K.	161 – 178, 317, 383
Schimel, D.	15
Schlesinger, D.	213
Schrefl, M.	433 – 446
Seifert, A.	115 – 133
Sofu, T.	349 – 361
Sperber, D.	115 – 133
Sperling, S.	219

Spivey, R.	135 – 149
Sreenivas, K.	325 – 338
Storms, B.	161 – 178, 317
Stalnov, O.	115 – 133
Taubert, L.	105 – 113
Tian, Y.	151 – 160
Van Raemdonck, G. M. R.	299 – 310
van Tooren, M. J. L.	299 – 310
Visser, K. D.	465
Watkins, S.	101, 387 – 402
Weber, D.	349 – 361
Whitfield, D.	325 – 338, 339 – 348
Willemsen, E.	311 – 315
Wong, A.	467
Wynnanski, I.	105 – 113
Yates, R. G.	259 – 297
Zhang, S.	243 – 258
Zhao, J.	239, 243 – 258