

Combined Analysis of Cooling Airflow and Aerodynamic Drag for a Class 8 Tractor Trailer Combination

2011-01-2288

Published
09/13/2011Raja Sengupta
Volvo Trucks North AmericaJeremy Beedy and Kevin Horrigan
Exa Corporation

Copyright © 2011 SAE International

doi:[10.4271/2011-01-2288](https://doi.org/10.4271/2011-01-2288)

ABSTRACT

Long haul tractor design in the future will be challenged by freight efficiency standards and emission legislations. Along with any improvements in aerodynamics, this will also require additional cooling capacity to handle the increased heat rejection from next generation engines, waste heat recovery and exhaust gas recirculation systems. Fan engagement will also have to be minimized under highway conditions to maximize fuel economy. These seemingly contradictory requirements will require design optimization via analysis techniques capable of predicting both the aerodynamic drag and engine cooling airflow accurately. This study builds on previous work [1] using a Lattice Boltzmann based computational method on a Volvo VNL tractor trailer combination. Simulation results are compared to tests conducted at National Research Council (NRC) Canada's wind tunnel. Correlation between simulation and experiment is presented for a half scale model, including the overall drag, surface pressure at locations over the truck outer surface and anemometer data collected behind the grille opening. Correlation for anemometer data is also presented for a full scale production truck in the same wind tunnel. The full scale production truck was then simulated in an on road environment to understand the impact of the wind tunnel environment on engine cooling airflow. Finally, the simulation results were analyzed to understand the limitation of using scale model tests to evaluate engine cooling airflow.

INTRODUCTION

Highway tractors are faced with multiple, somewhat conflicting demands. Rising fuel prices, and anticipated fuel economy standards are forcing manufacturers to focus on tractor trailer aerodynamics. Meanwhile, exhaust gas recirculation (EGR), employed to meet past emission legislations, has significantly increased engine cooling demand. In addition, future engine technologies, like waste heat recovery, are expected to add more heat load to the engine cooling system, if not additional heat exchangers. However, if these developments increase the amount of time the fan has to operate (also known as fan engagement) under highway conditions, that could lead to an overall decline in fuel economy.

Apart from its impact on fan engagement, the airflow through the grille also has a direct effect on aerodynamic drag, and therefore fuel economy. This is because the airflow splits into three parts: flow around the sides and top of the hood, flow under the vehicle, and flow through the grille. Since the overall aerodynamic drag is determined by the airflow over the entire vehicle surface, changes in this airflow split will inevitably influence the vehicle drag.

Finally, there is yet another way the airflow into the cooling package can influence fuel economy and that is via the Charge Air Cooler (CAC). Reduced airflow to the CAC will result in higher combustion air temperatures, thereby reducing the fuel efficiency of the engine. Therefore, it is critical to consider the engine cooling airflow needs and

Table 1. Summary of Configurations Studied

ID	Configuration Name	Geometry Detail	Scale	Trailer	Environment	Floor Condition	Wheels	Grille Config.
1	Half Scale Model, Open Grille	Test Model	0.5	14'	Wind tunnel	Stationary	Static	Open
2	Half Scale Model, Closed Grille	Test Model	0.5	14'	Wind tunnel	Stationary	Static	Closed
3	Full Scale Truck in WT	Production	1.0	28'	Wind tunnel	Stationary	Static	Open
4	Full Scale Truck in On road	Production	1.0	28'	On road	Moving	Rotating	Open

minimize fan engagement along with aerodynamic drag for long haul applications.

There are few tools available to obtain this information simultaneously. Cooling performance tests are typically carried out in chassis dynamometers, at low ram airspeeds and with fan fully engaged. While Wind Tunnel tests that are mainly focused on aerodynamic drag seldom measure airflow through the grille.

Moreover, it is expensive, if not difficult, to instrument the truck in a Wind Tunnel to measure airflow through the radiator. Even if microprobes are placed right before the radiator, without the engine running they would register higher airflow than expected, since active heat exchangers are more restrictive. It is also insufficient to simply consider the radiator circuit, since the CAC produces a thermal imprint on the radiator that affects the airflow through the entire cooling package and the overall cooling performance.

This is where Computational Fluid Dynamics (CFD) offers a unique opportunity. It is possible to simulate a tractor trailer combination at highway speeds, including an active engine cooling system with the fan engaged or disengaged. However, to gain confidence in such an approach, it has to be correlated against a feasible test. This provided the motivation for the anemometer airflow correlation presented here.

This work is a continuation of the SAE paper presented last year [1] which focused on the aerodynamic drag prediction and correlation with Wind Tunnel test data. The full scale and half scale tests also used anemometers to measure the airflow entering the engine bay. Correlation between simulation and test is presented for anemometer airflow rates, along with differences between the Wind Tunnel results and “on-road” simulation. Furthermore, it is shown that the radiator airflow map differs significantly from the distribution measured just behind the grille by the anemometers.

OVERVIEW OF CONFIGURATIONS STUDIED

As described in [1], experimental testing was conducted on both a half-scale Volvo VNL model as well as a full-scale production truck. [Table 1](#) summarizes the different models

and parameters varied for this study. Configurations 1 through 3 were selected to compare simulation results with wind tunnel measurements, investigate the influence of closing the grille on aerodynamic drag, and also evaluate the influence of geometric detail on under-hood cooling airflow. Configuration 4 was added to compare simulation results of cooling airflow between the wind tunnel and “on road” test environments.

SIMULATION APPROACH

The simulations in this study were carried out using a special discretization of the Lattice Boltzmann method (LBM) on a variable resolution Cartesian volumetric grid, which is automatically generated as part of the simulation process. In the LBM method, the fundamental equation of motion for fluid particles, the Boltzmann equation, is solved on a cubic lattice using a discrete set of fluid velocity states. The discrete Boltzmann equation is coupled with the ideal gas equation of state to represent realistic transfer of mass and momentum sufficient to describe small-scale fluid motions with very low numerical dissipation [2,3]. The Lattice Boltzmann method recovers the Navier-Stokes equations by tracking the collisions of particles, represented as particle density functions, conserving mass, momentum and energy for compressible flow [4]. The surface geometry is provided as an input in the form of a surface mesh with triangular facets. The geometry of the surface enters the LBM discretization as boundary facets immersed in the lattice, and they provide boundary conditions for the discrete velocity states by enforcing zero-flux through the wall.

A key feature of the LBM solver is that it is inherently time-dependent. Based on local and instantaneous flow field information, a local effective relaxation time in the Boltzmann collision is used to account for the unresolved universal scales of turbulent motion, similar to the introduction of eddy viscosity in a traditional Navier-Stokes solver [5]. Out of the three basic categories of turbulent scales of motion, i.e., the dissipative range, the inertial range and the anisotropic range, only the dissipative and inertial ranges lend themselves to a typical theoretical description assuming isotropy. The anisotropic turbulence associated with the largest scales of motion is not captured well by existing Reynolds Averaged Navier Stokes (RANS) models such as k-epsilon. Studies on tractor-trailer configurations

Table 2. Summary of Wind Tunnel Test Conditions

	Nominal Re (based on trailer width)	Nominal Velocity	Blockage Ratio at 0deg yaw	Characteristic Truck Area
Half Scale Model Tests	4.5×10^6	51.8 m/s	3.2%	2.67 m^2
Full Scale Tests	4.5×10^6	25.9 m/s	12.9%	10.68 m^2



(a)



(b)

Figure 1. Half-scale Model truck mounted in NRC wind tunnel (a) Experiment (b) Simulation

have shown deficiencies of RANS in accurately predicting pressure distributions in the trailer base region which is characterized by large scale, transient structures [6]. The approach adopted in this study uses turbulence theory to model only the dissipative and inertial ranges (unlike most approaches), modeling the dynamics of sub-grid scale turbulence by two additional equations derived from an extended Renormalization Group theory (RNG) formulation. This approach is known as Very Large Eddy Simulation (VLES) [7]. This VLES formulation has been confirmed to show low grid sensitivity of the time-averaged flow results when the most energetic turbulent scales have been resolved [8,9]. The assumption of fully turbulent flow is reasonable because a full scale truck at highway speeds does not have regions of laminar flow which contribute significantly to the overall aerodynamics of the vehicle.

WIND TUNNEL ENVIRONMENT

Experimental work for this study was conducted at the National Research Council (NRC) of Canada 9×9 m wind tunnel located at the Upland Campus (Ottawa, Canada). The facility and the test setup are described in detail in [1]. The numerical simulations for Configurations 1-3 from Table 1 incorporated many of the wind tunnel environment details that have been shown to influence aerodynamic results, including geometry of the wind tunnel walls, the mounting hardware to the wind tunnel floor, and modeling of the

boundary layer suction system. A summary of the test flow conditions is shown in Table 2. The wind tunnel results presented here are uncorrected for blockage since the focus was primarily on airflow rates. Aerodynamic drag comparisons are presented for open vs. closed grille, but only for the half scale model, at a fixed blockage ratio.

For the scale model tests, a moving belt system was used to mimic a moving ground condition and the tractor wheels were driven at the same speed as the belt. The boundary layer forming upstream in the wind tunnel was removed using a boundary layer suction system [10]. The moving belt system was simulated by applying a moving wall boundary condition to the area of the belt. The boundary layer suction system was modeled by using a frictionless floor condition upstream of the final suction area, and a standard frictional wall boundary condition between the downstream location of the final suction system and the moving belt starting location. This allowed the development of a boundary layer upstream of the moving belt, both in the experiment and the simulation. A friction floor boundary condition was applied to the floor on either side of the belt, downstream of the final suction system location. In the simulation setup, rotational wall boundary condition was applied to the wheels that were driven in the experiment (one steer axle and two drive axles of the tractor). For full scale wind tunnel tests, the boundary layer suction system was turned off. In the simulation, a frictional wall



(a)



(b)

Figure 2. Full-scale truck mounted in NRC wind tunnel (a) Experiment (chassis fairings removed) (b) Simulation

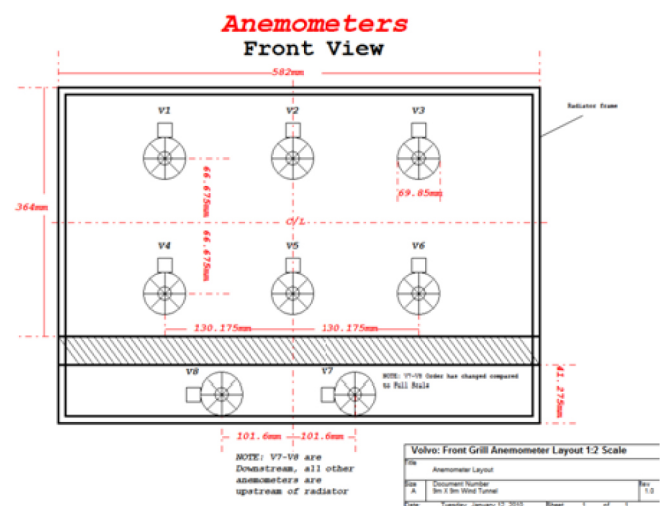


Figure 3. Half-scale model anemometer mounting locations. Anemometers are numbered 1-8 from top left to bottom right

boundary condition was applied to a region upstream of the turntable in order to reproduce the boundary layer height that was measured at the leading edge of the turntable in the empty tunnel. This upstream distance was calculated based on the measured boundary layer height using flat plate boundary layer theory. All the mount plates, mount cradles, and air-bearings were also included in the simulation in order to match the experimental ground clearance and also model the flow effects accurately in this region. [Figures 1](#) and [2](#) show images of the truck models in the wind tunnel for experimental and simulation setups.

ANEMOMETER MEASUREMENTS

During wind tunnel testing of the half-scale model, velocity measurements were taken between the grille and front face of

the mesh screen used to represent the radiator restriction. Eight anemometers were placed in the underhood region. Due to space constraint, only six could be mounted in front of the screen used to represent the cooling module restriction. The remaining two anemometers were mounted behind the screen at the bottom (see [Figure 3](#)). Similar airflow measurements were taken during the full-scale model test using anemometers mounted entirely in front of the cooling module (see [Figure 4](#)).

ON ROAD ENVIRONMENT

Configuration 4 from [Table 1](#) was modeled using a simulation of the “On road” environment. This included a very low blockage computational domain, a full moving ground plane relative to the truck, and a rotating wall