

Comparative Studies between CFD and Wind Tunnel Measurements of Cooling Performance and External Aerodynamics for a Heavy Truck

Helena Martini
Chalmers Univ. of Technology

Peter Gullberg
Volvo Group Trucks Technology

Lennart Lofdahl
Chalmers Univ. of Technology

ABSTRACT

Nowadays, much focus for vehicle manufacturers is directed towards improving the energy efficiency of their products. The aerodynamic drag constitutes one major part of the total driving resistance for a vehicle travelling at higher speeds. In fact, above approximately 80km/h the aerodynamic drag is the dominating resistance acting on a truck. Hence the importance of reducing this resistance is apparent. Cooling drag is one part of the total aerodynamic drag, which arises from air flowing through the heat exchangers, and the irregular under-hood area.

When using Computational Fluid Dynamics (CFD) in the development process it is of great importance to ensure that the methods used are accurately capturing the physics of the flow. This paper deals with comparative studies between CFD and wind-tunnel tests.

In this paper, two comparative studies are presented. One is a comparison between cooling performance simulations and chassis dynamometer measurements; the other study is a comparison between external aerodynamics simulations and wind-tunnel measurements.

The purpose of this study was to evaluate and develop methods and models for determining aerodynamic drag and cooling performance, which ultimately will be used to develop more energy-efficient cooling of heavy trucks.

The results from the two comparative studies showed that there was in general good agreement between CFD and wind-tunnel measurements. For the cooling performance simulations, the analysed parameters were very close to the measured values. For the external aerodynamics simulations, the results were not easy to analyse. The overall results were still satisfactory; for the simulated yaw angles, the drag coefficient (C_D) values were less than 4.1% different from measured data.

CITATION: Martini, H., Gullberg, P., and Lofdahl, L., "Comparative Studies between CFD and Wind Tunnel Measurements of Cooling Performance and External Aerodynamics for a Heavy Truck," *SAE Int. J. Commer. Veh.* 7(2):2014, doi:10.4271/2014-01-2443.

INTRODUCTION

Reducing the harmful effect of vehicles on the environment is a key question for vehicle manufacturers today. Constantly increasing oil prices and stricter emission legislations are also important factors for the vehicle industry to consider. There are many factors affecting the fuel consumption of a vehicle. One of the major ones is the aerodynamic resistance. In fact, above 80km/h, the aerodynamic drag is the most significant

resistance for a truck driving at constant speed, according to Löfdberg [1]. Hence the importance of working with the aerodynamic properties of trucks is apparent.

Cooling drag is a part of the total aerodynamic drag, arising as a consequence of air flowing through the grille, cooling module and the irregular under-hood area. Hallberg [2] states that the trend is towards more and more cooling being required not only as a consequence of increased engine power, but also for

the Air Conditioning (AC) system, exhaust after-treatment systems and auxiliary systems to function properly. Hallberg [2] further states that the cooling-drag component can be up to 8% of the total drag, which is a substantial amount. At the same time as the cooling demand is increasing, there are also discussions regarding how to be able to close off some cooling flows, for example at highway speeds, when the full cooling capacity is not needed. In such situations, fuel consumption may be lowered as the cooling airflow is cut off.

When evaluating the aerodynamic performance of a vehicle, CFD and wind-tunnel measurements are often used together. Regarding CFD, it is of great importance that the simulations actually capture the physics of the fluid flow. This is a significant challenge, and wind-tunnel measurements here play an important role in judging the accuracy of CFD.

A large number of correlation studies have been published during the recent years. Horrigan et al [3] presented a study where CFD simulation results were compared to wind-tunnel measurements from University of Washington Aeronautical Laboratory (UWAL). The results from the work showed that CFD under-predicted the drag coefficient by 6.5% compared to wind-tunnel data. The authors discussed difficulties with the simulation model and also cited: the wind-tunnel geometry, matching the inlet flow conditions, and to properly model the pressure boundary condition at the side vents, as possible sources of error.

Heinecke et al [4] evaluated the aerodynamic drag from CFD and wind-tunnel measurements in the National Research Council (NRC) tunnel in Canada for a production Volvo VNL tractor-trailer combination. For a full-size tractor-trailer combination in the NRC tunnel, the blockage effect was significant. The authors also showed that simulation results were within 2% of wind-tunnel data, for three simulated yaw angles.

Sengupta et al [5] performed a correlation study between CFD and wind-tunnel measurements in the NRC wind tunnel for a Volvo VNL tractor-trailer combination. Parameters that were correlated in the study for the half-scale model were the overall drag values, surface pressure along the truck surface and anemometer data collected from behind the grille opening. In addition, anemometer data for the full-size model were also added to the investigation. The authors presented a calculated percentage of 1% cooling drag for the half-scale model, according to CFD. This value was consistent with wind-tunnel measurements. Centre-line pressure distribution results showed generally good agreement between CFD and wind-tunnel data. Concerning the results of the cooling airflow, the velocity distribution over the radiator surface showed different results for the half-scale model and the full-size model. The authors discussed this phenomenon and concluded that there were several discrepancy sources, among them the geometry of the under-hood of the half-scale model and only measuring the velocity at eight discrete points.

Lafferty et al [6] performed an investigation where CFD simulations and cooling cell measurements were compared, to validate the computational method. The results from the analysis showed that the difference in radiator Top Tank Temperature (TTT) between CFD and test data was 2.9°C and 1.2°C, for two different operating points of 1200rpm and 1600rpm, respectively. These results were obtained after a correction for the porous media coefficients.

Bhatnagar et al [7] evaluated the cooling airflow for a Freightliner Class 8 truck, with focus on the radiator TTT values and the Charge Air Cooler (CAC) outlet temperatures. The results from the study showed that there was good agreement between CFD and measurements; the TTT values were found to vary between 0.3°C and 3.2°C from measurements, while the CAC outlet temperatures were determined to be 0.4°C above and 0.1°C below measured data.

Alajbegovic et al [8] performed a similar study, but for a Land Rover LR3 and a Ford Mondeo. For the radiator TTT values, the largest deviation from measurements were 4.2°C, most of the results were 2.3° or lower. The authors also conclude that their combined method for determining cooling airflow and aerodynamic drag was successful.

In this paper, two comparative studies are presented and discussed. The first study compares results from cooling performance simulations to chassis dynamometer tests. The second study compares drag coefficient values from CFD with wind-tunnel measurements. The overall objective with the study was to generate more knowledge in the modelling of aerodynamics and cooling performance; to gain a deeper understanding of suitable models and methods for these kinds of simulations. The results are to be used for upcoming investigations, with more energy-efficient cooling for heavy trucks as the main target.

The aim of this study was to run CFD simulations and compare to test data, and evaluate the methods of these simulations. Cooling performance simulations were performed to compare to measured data from climatic wind-tunnel tests performed in the Volvo Vehicle Feature Laboratory (VFL), and external aerodynamics simulations were run with the results being compared to measured data from the NRC wind tunnel in Canada [9]. It was of interest to ensure that the difference in analysed parameters compared to measured data was of a satisfactory level.

TEST OBJECTS

In this section, the vehicle models and the different modifications of the models are presented.

Cooling Performance

The vehicle model used for the cooling performance analysis was a full-size Volvo FH 4x2 tractor unit with a box-shaped body. There was no actual trailer fitted, but rather a box-shaped body.

was placed on the chassis of the tractor, to resemble the vehicle set-up during the VFL measurements. The vehicle geometry can be seen in [Figure 1](#).



Figure 1. Vehicle geometry used for the cooling performance simulations.

External Aerodynamics

The vehicle model used for the external aerodynamics analysis was a full-size Volvo FH 4x2 tractor unit. The vehicle was equipped with a full aerodynamic package, including roof deflector, cab side extenders and chassis skirts on the tractor unit. The length of the trailer model used was 8.52m ($\approx 28'$), which is considerably shorter than trailers seen on the road for this type of vehicle combination. However, this trailer model was used since it corresponded to the trailer model used in the wind-tunnel measurements.

For the external aerodynamics simulations, two different vehicle geometry set-ups were tested, the first set-up being a reference model with the grille open, the second being a model with an entirely sealed front. These two configurations were chosen in order to not only compare the absolute levels of the drag coefficients between CFD and wind-tunnel measurements, but also to investigate if the delta-values between two configurations were consistent. The reference vehicle geometry can be seen in [Figure 2](#) and the sealed-front model is displayed in [Figure 3](#).



Figure 2. Geometry of the reference model for the external aerodynamics simulations.

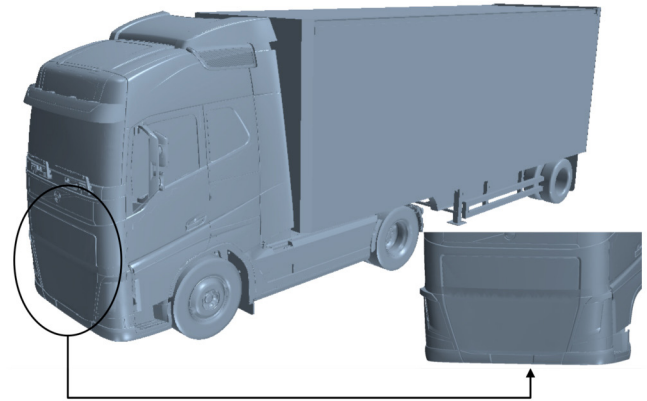


Figure 3. Geometry of the model with sealed front, seen in a close-up in the bottom right corner.

NUMERICAL SET-UP

The investigations performed in this paper were performed using CFD. The Computer Aided Design (CAD) cleaning of the vehicle geometry and initial surface meshing was done using ANSA by Beta CAE Systems [10], while the vehicle model was wrapped, surface meshed, volume meshed and run in STAR-CCM+ by CD-adapco [11].

Cooling Performance

For the cooling performance simulations, the vehicle was simulated in the Volvo VFL tunnel, to be able to compare with test data. The simulation set-up of the vehicle model and wind-tunnel domain can be seen in [Figure 4](#).

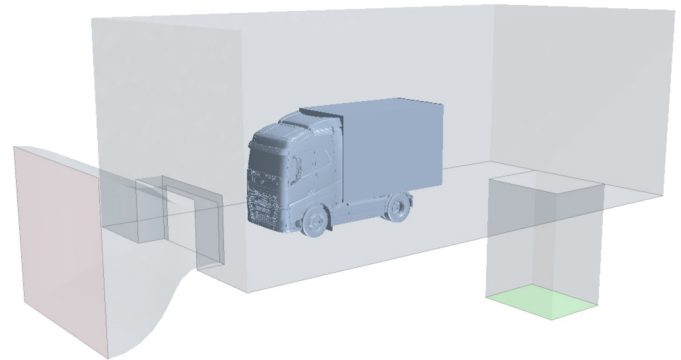


Figure 4. Overview of the simulation set-up in the Volvo VFL tunnel.

Mesh

The volume mesh was hexahedral dominant, using the STA-RCCM+ *Trimmer* model [12]. On the vehicle surface, one prism layer with a height of 1mm was created. The only exception was the cooling fan, which had 8 prism layers with a total height of 3mm.

The volume meshes for the cooling performance simulations were identical for all tested operating points in the study.

The volume mesh for the cooling performance simulations is displayed in [Figure 5](#).

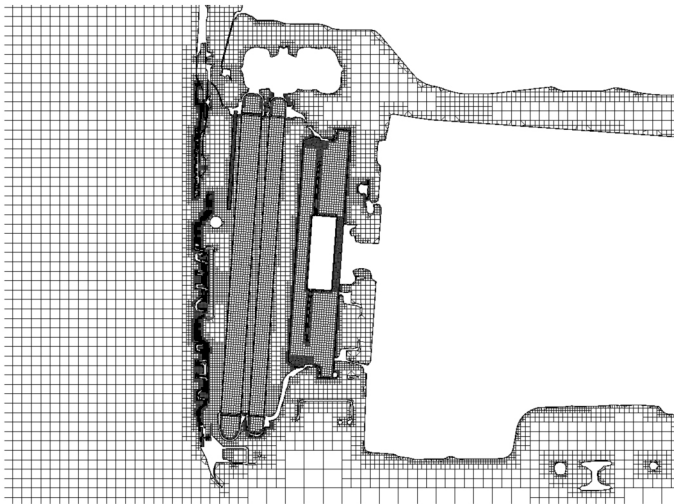


Figure 5. Overview of the volume mesh for the cooling performance simulations, near the vehicle surface. The plane is cut along the vehicle centre-line and the mesh density in the under-hood region and fan region can be seen. Note that the entire calculation domain is not visualized.

Simulations

The simulations were carried out based on a steady Reynolds-Averaged Navier-Stokes approach. To model the turbulent properties of the flow field, the realizable $k\epsilon$ *Two-Layer* model was used. The *Two-Layer approach* is a method offering mesh flexibility by switching between a High-Reynolds Number approach and a Low-Reynolds Number approach, depending on the resolution of the mesh near the surface. Also, within the buffer region, where $1 < y^+ < 30$, this *Two-Layer approach* was expected to produce the most reliable results [12].

Fan Model

The Multiple Reference Frame (MRF) model was used to model the fan rotation in the simulations. This model is a frozen rotor method. The user defines a control volume around the fan, and the model simulates the effect of rotation on a stationary mesh inside this control volume. The size of the control volume, or MRF region, is a sensitive parameter for the accuracy of the results. The choice of MRF region size was made in accordance with results from Gullberg et al [13].

Heat Exchanger Model

The heat exchanger surface was modelled as porous media, in order to avoid the extremely large meshes that would be necessary to resolve the entire structure of a detailed heat exchanger.

The porous media formulation includes parameters, which define the effects of the porous media on the flow field downstream the heat exchanger. The parameters are included in source terms in the momentum equation. The parts that should be modelled as porous media are assigned their own flow region, where coefficients of flow resistance are added. These coefficients are then used to calculate the porous source term in the momentum equation [12].

The porous source term is defined as in [Equation 1](#).

$$f_p = -(P_v + P_i |v|) \cdot v \quad (1)$$

In [Equation 1](#), P_v and P_i are the viscous and inertial resistance tensors, respectively. The values of the inertial and viscous resistance are obtained by pressure drop measurements over each heat exchanger component and the values were obtained from the manufacturer.

To include cooling performance analysis, a heat exchanger model was implemented in the simulation. The STAR-CCM+ *Actual Dual Stream* heat exchanger model [12] was used to evaluate the heat exchange between the hot and cold cores. The hot core is defined as part of the heat exchanger where the coolant and charge air is flowing, whereas the cold core is defined as the part where air passes through the heat exchanger to cool the coolant and charge air. The heat exchanger model also needed information about the specific heat exchanger components that were used. Files including the mass flow through the hot and cold cores, and the corresponding heat-transfer rate were therefore provided to the simulation model.

Operating Conditions

The cooling performance was evaluated at four different operating points, all of them critical for establishing sufficient cooling performance. The engine speeds of these operating points were 1100rpm, 1300rpm, 1500rpm and 1800rpm. All four operating points are steady state points that are commonly used for characterizing the cooling performance. The operating points have different input values in terms of mass flow of air into the Charge Air Cooler (CAC), target heat rejection of the radiator, volume flow of coolant in the radiator, fan speed, and the inlet temperature of the charge air into the CAC.

Analyzed Parameters

In this section, a brief explanation of the analyzed parameters for the cooling performance simulations is presented. These parameters were used to compare the measurements with the CFD results.