

Experimental and Numerical Investigation of Wheel Housing Aerodynamics on Heavy Trucks

David Söderblom
Chalmers Univ. of Technology

Per Elofsson
Scania AB

Linus Hjelm
Volvo 3P

Lennart Lofdahl
Chalmers Univ of Technology

ABSTRACT

Wheel and underbody aerodynamics have become important topics in the search to reduce the aerodynamic drag of the heavy trucks. This study aims to investigate, experimentally as well as numerically, the local flow field around the wheels and in the wheel housing on a heavy truck; and how different approaches to modelling the wheel rotation in CFD influences the results. Emphasis is on effects due to ground simulation, and both moving ground and wheel rotation were requirements for this study.

A 1:4-scale model of part of a heavy truck geometry has been developed. During the model design numerical simulations were used to optimise the shape, in order to replicate the flow field near the wheel of a complete truck. This was done by changing the flow angles of the incoming and exiting flows, and by keeping the mass flow rates in to, and out of, the wheel housing at the same ratios as in a reference full size vehicle. To reduce blockage effects, the model was sectioned to reduce both height and width. In the experiments, pressure sensors and static pressure taps located in the wheel housing were utilised, and the simulations replicated the boundary conditions of the wind tunnel experiments, both in terms of the geometry of the model and wind tunnel as well as the modelling of the ground simulation.

It was found that the wheel wake structures changed significantly when ground simulation was utilised. The main outflow through the wheel housing was influenced by the wheel rotation and took place further upstream, which resulted in large differences in the flow field downstream of the wheel. The influence of different strategies for modelling the wheel rotation in CFD was investigated and it was found that the Sliding Mesh approach was the most accurate method.

CITATION: Söderblom, D., Elofsson, P., Hjelm, L. and Lofdahl, L., "Experimental and Numerical Investigation of Wheel Housing Aerodynamics on Heavy Trucks," *SAE Int. J. Commer. Veh.* 5(1):2012, doi:10.4271/2012-01-0106.

INTRODUCTION

Today focus is on reducing the fuel consumption and emission levels of heavy trucks. The aerodynamic drag becomes the dominating driving resistance at about 80km/h for a typical 40 ton tractor-trailer combination, and since most European long-haulage trucks operate around 90km/h a reduction of the aerodynamic drag will have a significant influence on the fuel consumption. The aerodynamic shape optimisation of heavy trucks has mainly focused on

improving the cab shape and adding drag reducing devices to the tractor unit. However, the underbody, including wheel housings, wheels and engine compartment constitutes a substantial part of the total aerodynamic drag, and for some vehicle configurations can account for as much as half of the total aerodynamic drag.

In order to improve these parts of the vehicle, the boundary conditions must correspond to the reality, that is, there is a need to simulate the relative ground motion and wheel rotation. The influence of ground simulation (moving

ground and rotating wheels), has been thoroughly investigated on passenger cars considering both the aerodynamic forces and flow field [1, 2, 3, 4, 5]. It was seen that the drag coefficient tended to decrease with moving ground and rotating wheels, and that the whole flow field, not only locally, was affected by the ground simulation. Elofsson and Bannister [3] have shown that strong interference effects between the rear wheel wakes and the base wake could be observed. Wickern et al. [4] showed that the influence of ground simulation on the measured drag varied for different types of vehicles, and it was shown that this was related to how the ground simulation affected the up- or downwash behind the vehicle. Studies on heavy trucks [6 and 7] have shown that the wind averaged drag coefficient tended to increase with ground simulation. There are only a few studies [8 and 9] that have analysed the influence on the flow field when using moving ground and rotating wheels for heavy trucks, and none of them presented any experimental results regarding the flow field.

The aim of this study was to perform experiments and simulations on a heavy truck model and analyse the flow under the influence of moving ground and wheel rotation. The tractor-trailer combination under consideration has been used in previous studies by the author but in order to achieve detailed flow field measurements some new concepts had to be evaluated. It was concluded that the scale had to be kept larger than 1:4, but this would result in excessive blockage in the wind tunnel under consideration. Therefore the model had to be reduced in both height and width to minimise the blockage level. This resulted in a wind tunnel model consisting of the front left corner of a truck with a fully detailed wheel and wheel housing, that was equipped with a calibrated heat exchanger model as well as a representation of the engine. Similar models representing only a part of a vehicle have been seen in studies by Riederer [10] and Dreyer et al. [11]. The design of the model was performed using numerical simulations. The initial simulations showed that there was no advantage of having a symmetry plate mounted in the test section of the wind tunnel, and that a number of uncertainties would arise due to the interference effects with that kind of plate. In order to replicate the flow around the wheel of a tractor-trailer combination a number of important parameters were studied. These were: the angles of the flow passing ahead and aft of the wheel, the mass flow rates through the heat exchangers, and the mass flow rate through the wheel housing opening. Several compromises had to be made in order to get a rigid model that gave reliable results, instead of having a geometry that reproduced the correct flow conditions but was unstable at the considered Reynolds number. The blockage level of the final geometry, when considering the test section of the Chalmers wind tunnel, was 7.8 %. For more details regarding the development of this model see the work by Söderblom [12].

This paper aims to present an analysis of the flow field around the wheel of a heavy truck model using both experimental and numerical methods. Two configurations

were considered for this paper, and these were to use different boundary conditions for the ground simulation, that is, stationary ground, or moving ground with wheel rotation. The influence on the drag coefficient was not investigated since it was not possible to measure the aerodynamic forces acting on the wheel. Crosswind conditions are of utmost importance for heavy trucks but these could not be taken into account in this study, since the model used only consisted of the front left corner of a truck; and also there was no possibility to yaw the moving ground system and model support.

METHODOLOGY

EXPERIMENTAL SET-UP

The Chalmers low speed wind tunnel L2 is a closed-loop wind tunnel, with a closed test section that measures $1.25 \text{ m} \times 1.8 \text{ m} \times 3 \text{ m}$ ($H \times W \times L$). The maximum speed of the wind tunnel is 63 m/s, and it has a turbulence intensity level of 0.1%. It is equipped with a moving ground system of single-belt type that is 975 mm wide $\times$ 1725 mm long. Boundary layer suction was applied through a perforated plate upstream of the moving belt. With boundary layer suction and stationary ground conditions the displacement thickness was 1.8 mm in a position 10 mm downstream of the suction plate, and it grew to 3.7 mm 250 mm downstream of the first measurement point [13]. When ground simulation was utilised the displacement thickness was 1.8 mm at the point 10 mm after the suction plate and it was 1.8 mm in the downstream position as well. With the moving ground system placed in the wind tunnel the vehicle models have to be suspended by a carbon-fibre sting from the top of the test section.

A 1:4-scale model of a heavy truck geometry was used. The model consisted of the front left corner of the truck as described in the previous chapter. It was suspended by a vertical carbon-fibre strut and the wheel was mounted on a separate axle in order to reduce the risk that the wheel would bounce or slip on the belt. The axle was extended through the vehicle body in order to have undisturbed flow close to the wheel and wheel housing. All the experiments in this study were run at 35 m/s corresponding to a Reynolds number of approximately 910 000 based on the maximum width [14]. The model can be seen in figures 1 and 2.

The body parts were manufactured by rapid prototyping and these parts were then bolted onto the chassis of the model. The chassis was built up from steel profiles and machined aluminium parts which were bolted together. A representation of the cooling package was designed and manufactured. It consisted of several fine wire meshes to give the correct pressure drop over the heat exchangers, and honeycomb for its flow straightening properties. The model radiator was then calibrated in the same manner as presented by Christoffersen et al. [15]. The wheel was based on the geometry used in the numerical simulations in this project, and was manufactured in aluminium. It was then balanced

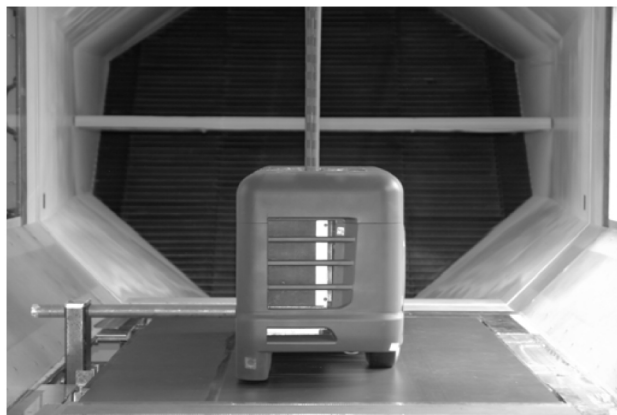


Figure 1. Front view of the truck model mounted in the Chalmers wind tunnel L2.



Figure 2. The truck model mounted in the Chalmers wind tunnel L2.

and the tread surface was coated with a thin layer of rubber in order to reduce vibrations, and to make sure that there was enough friction to have the wheel driven by the moving belt.

The model was equipped with 61 surface pressure taps, 39 of these were located in the wheel housing area. The remaining pressure taps were found in the base of the vehicle, in the stagnation area and also to monitor the pressure drop over the cooling package. The taps were connected to two Pressure Systems Int. DTC 32-port pressure scanners that were mounted inside the model. The sampling rate was set to 100Hz and 256 samples were taken at each time. This was repeated five times at 50 second intervals for each configuration and the data was then averaged for the post-processing. The reference pressure for the pressure scanners was fed through the carbon-fibre sting that supported the model. The pressure taps were connected to the pressure scanners with polyester tubes having an internal diameter of 1mm.

For the flow field measurements a seven-hole probe was used. With such a probe it is possible to measure local total and static pressures and flow angles up to 70 degrees relative to the probe [16]. The probe was connected to a Pressure

Systems Int. 9116 pressure scanner and the sampling rate was set to 62.5 Hz and 250 samples were taken at each measurement point. A traversing gear was mounted on top of the test section and enabled the probe to travel 400 mm in both X- and Z-directions. The span-wise position (Y-direction) had to be set manually, and the range, including two different probe mounts, was 240 mm.

NUMERICAL SET-UP

Modification of the geometry and the surface mesh was carried out using the software ANSA, and the case set-up was performed in EXA PowerCASE 4.3b. In the simulations the Chalmers low speed wind tunnel L2 was replicated, that is, an identical test section, contraction and diffuser was included. In order to reduce the risk of transient waves travelling through the domain in the initial stage of the simulations the diffuser and settling chamber was extended two hydraulic diameters downstream and upstream according to recommendations from the software vendor. The walls were modelled as standard wall, and the part of the floor that represented the moving ground system and free stream velocity were set to 35m/s. The inlet was defined as a mass flow inlet and the outlet as a mass flow outlet. Since the wind tunnel has a breather slot downstream of the test section, this gap had to be modelled. This was done by including the gap and adding a large external volume surrounding the closed test section. The size of the volume was $4\text{ m} \times 6.5\text{ m} \times 3\text{ m}$ ($H \times W \times L$). In [figure 3](#) the computational domain can be seen.

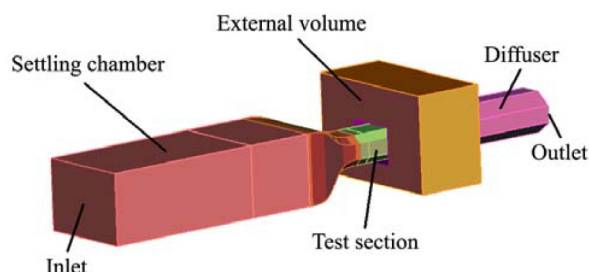


Figure 3. Computational domain used in the numerical simulations.

Three methods exist for modelling the wheel rotation. These are: boundary condition Rotating Wall, Moving Reference Frames (MRF), and Sliding Mesh. The Rotating Wall boundary condition behaves like an ordinary wall, but one that rotates around an axis [16]. In the MRF approach all solid objects within the reference frame with a no-slip boundary are automatically treated as rotating bodies. All voxels within the region are assigned by a rotational velocity and after the simulation the result are converted back to the global co-ordinate system by subtracting the rotational velocity. The Sliding Mesh approach physically rotates the geometry with respect to the local reference frame, and the information about the flow properties will be transferred over the boundaries of the local and global reference frames [17]. As well as this, one simulation was also carried out with stationary ground without wheel rotation.

EXA PowerFLOW 4.3b, a Lattice-Boltzmann based solver, was used for the simulations. These are always run transient and the physical time per time step is determined by cell size and velocity scale. For more details on this method see, for example [18] and [19]. For modelling the turbulence the k- ϵ RNG-model was used, which is the only model available. The model size was 37.4×10^6 voxels (14.9×10^6 fine equivalent) and 10.9×10^6 surfels (4.9×10^6 fine equivalent). The simulations were run on 128 CPU's for about 44 hours, and the simulated time was 0.7 seconds.

The model used in the simulations was based on the CAD data from the wind tunnel model, including the support sting and wheel axle, in order to have the identical geometry as shown in figure 4. This meant that all internal gaps and volumes were kept and the only significant difference was the lack of pressure tubes inside the model

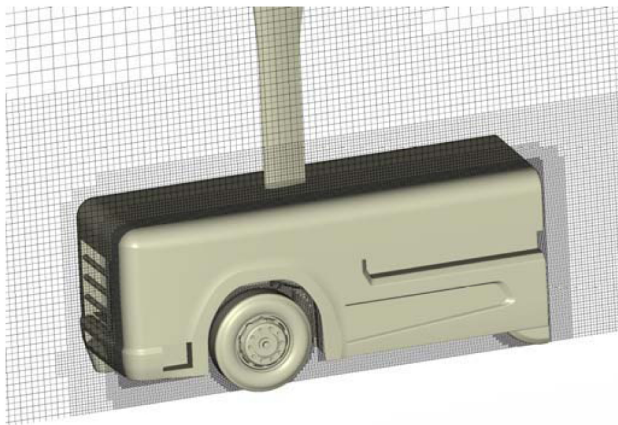


Figure 4. Model used in the numerical simulations and an example of grid refinement levels. Note that the entire domain is not shown.

The finest elements were located near the grille openings in order to fully resolve the porous media representing the mesh grilles. Eight refinement levels were used for the computational grid; the element size close to the wheel and in the wheel housing was 2 mm. The same size of voxels was used for the entire model including the support sting and wheel axle as can be seen in figure 4.

Several studies have been presented on how to model rotating wheels on passenger cars in CFD [5] and [20 - 21]. However, no published work on modelling heavy truck wheels is available. It was therefore decided as a part of this study to investigate the influence of wheel modelling for heavy trucks; since the geometry of a truck wheel is significantly different to the wheel of a passenger car. Since the design of the spokes have a particular influence on the flow through the openings of the wheel, and that affects the formation of the wheel wake, it was considered to be of interest to investigate the influence of different modelling strategies for heavy truck wheels as well. Three different approaches to model the wheel rotation was made and for the MRF method, two different choices of domain were investigated. The different local rotating reference frames are presented in figure 5. The two different approaches for the MRF method were based on recommendations in the software guidelines [17] (MRF I), and according to the recommendations for MRF in RANS by Wäschle [5] and Mlinaric [21] (MRF II): that is, the volume should only enclose the openings of the wheel. The Sliding Mesh domain was also defined according to guidelines from the software vendor [17].

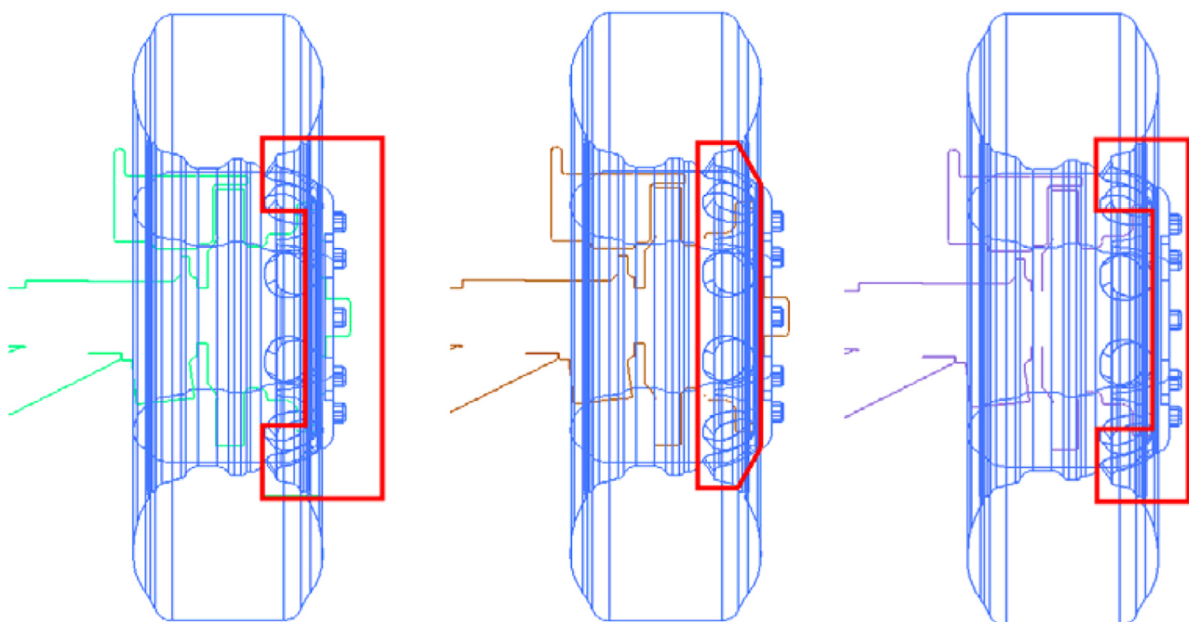


Figure 5. Choice of rotating reference frame. From left: MRF I, MRF II and Sliding Mesh.