

Investigation of Wheel Aerodynamic Resistance of Passenger Cars

Alexey Vdovin and Lennart Lofdahl
Chalmers Univ. of Technology

Simone Sebben
Volvo Car Corp.

ABSTRACT

There are a number of numerical and experimental studies of the aerodynamic performance of wheels that have been published. They show that wheels and wheel-housing flows are responsible for a substantial part of the total aerodynamic drag on passenger vehicles. Previous investigations have also shown that aerodynamic resistance moment acting on rotating wheels, sometimes referred to as ventilation resistance or ventilation torque is a significant contributor to the total aerodynamic resistance of the vehicle; therefore it should not be neglected when designing the wheel-housing area.

This work presents a numerical study of the wheel ventilation resistance moment and factors that affect it, using computational fluid dynamics (CFD). It is demonstrated how pressure and shear forces acting on different rotating parts of the wheel affect the ventilation torque. It is also shown how a simple change of rim design can lead to a significant decrease in power consumption of the vehicle. A way of introducing ventilation torque into the driving resistance equation is discussed.

The results show that it is possible to assess ventilation resistance moment using CFD. It is demonstrated that brake discs have almost negligible ventilation torque, while the contribution of rims and tyres may vary depending on the rim design used and the velocity of the vehicle. It is also shown that for the designs investigated the equivalent ventilation force has a second order dependency on the vehicle velocity which allows an introduction of a ventilation resistance coefficient $C_{D(vent)}$ that is independent of velocity.

CITATION: Vdovin, A., Lofdahl, L., and Sebben, S., "Investigation of Wheel Aerodynamic Resistance of Passenger Cars," *SAE Int. J. Passeng. Cars - Mech. Syst.* 7(2):2014, doi:10.4271/2014-01-0606.

INTRODUCTION

Around the world, constantly growing fuel prices and increasing environmental requirements for road vehicle emissions make it important for vehicle manufactures to address the problem of reducing the fuel consumption. The latter can be achieved by different means and one of the ways to do so is to improve vehicle aerodynamic design. This is one of the most effective ways since at speeds above 60-70 km/h aerodynamic drag force dominates all other resistance forces acting on a vehicle [1].

There are a number of regions of the vehicle that are known to be of particular importance for aerodynamic drag reduction. In this study the area investigated is wheels and wheel-houses. This region accounts for around 30% of vehicle aerodynamic drag [2], [3], [4], [5], [6] moreover there are also other effects in the area that are responsible for additional resistance that are usually left unaccounted for. There is an aerodynamic

resistance moment that acts on parts that are rotating in air, in this case the wheels. This moment is sometimes referred to as "ventilation drag" or "ventilation torque" [7].

This aerodynamic resistance moment is rather difficult to measure in the wind tunnel since it is hard to separate it from the rolling resistance. There are different methods described in the literature and all of them require a set of simplifications and geometrical modifications in order for the measurement to be made [8], [9].

CFD on the other hand offers an easier way to predict the ventilation torque since there is no rolling resistance calculation and the deformed shape of the tyre can be used. Of course, one should not forget that CFD has its own limitations and uncertainties, depending on the simulation method and boundary conditions used.

There are pressure and shear forces acting on any surface affected by the airflow. In the case of the rotating wheel, the surfaces affected are the tyre, the rim and the brake disc, see [Figure 1](#). The forces mentioned produce a rotational moment of resistance around the wheel axle of rotation - the ventilation torque, T_{vent} .

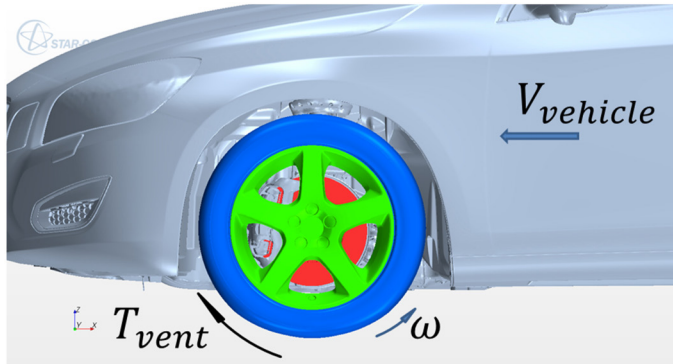


Figure 1. Rotating parts of the wheel

METHOD

The work presented in this paper is an analysis of results obtained from numerical simulations. The simulations and post-processing were conducted in the commercial CFD software StarCCM+ v8.04. For pre-processing purposes the commercial code ANSA v14.2.0 was used.

The Model

The model tested was a full-size Volvo S60. Detailed vehicle geometry was prepared in the pre-processing software, where part geometries were cleaned and put together. The vehicle ride height and pitch angle were adjusted to match the real on-road situation. Slick tyre models with production tyre profiles obtained from the manufacturers were morphed to correspond to the shape of the real loaded tyres working at 100 km/h [10]. Suspension geometry and engine compartment were represented with a high level of accuracy. The model can be seen in [Figure 2](#).

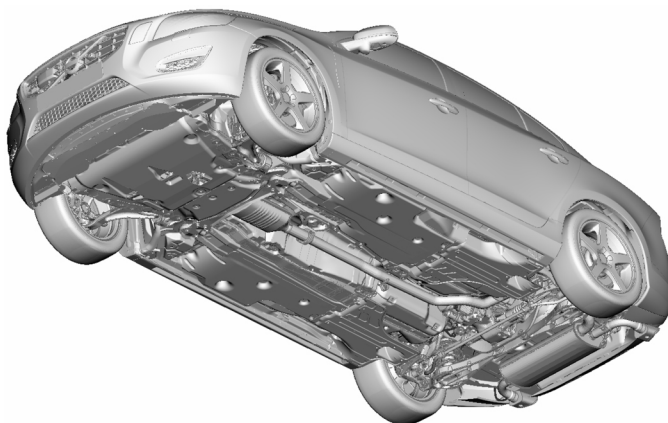


Figure 2. Vehicle geometry used for simulations

As a part of the preparation process a number of interface surfaces were introduced into the model in order to be able to separate regions with different physics specifications or for different volume mesh type desired.

Numerical Set-Up

The numerical method used was a standard automotive CFD procedure based on guidelines from the Volvo aerodynamic calculation practice. Due to the complexity of the vehicle geometry and especially the engine compartment area, it was required to run a surface wrapper that allowed fixing possible single-sided faces, gaps and intersecting elements. The surface mesh size used has a maximum length of 7 mm, being automatically refined down to 2 mm on the sharp edges and other geometrical features where this refinement was required. The volume mesh generated had 3 prism layers of total height of 7 mm in order to properly resolve boundary layer and maintain y^+ values in the range between 30 and 150 [11]. The mesh generated was mostly hexahedral with refinement zones for the engine bay, underbody, wheel-houses, mirrors and base wake areas, see [Figure 3](#).

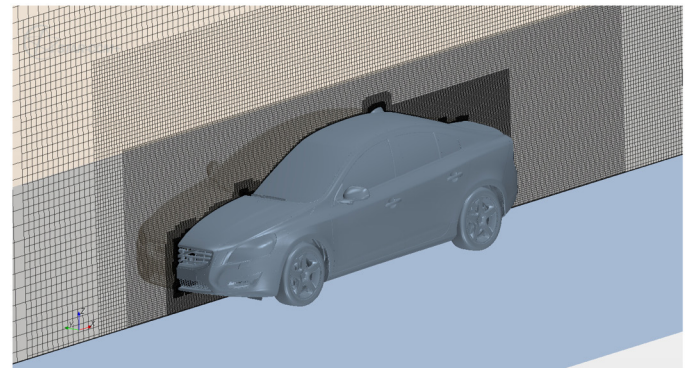


Figure 3. Volume mesh with refinement regions shown

The components of the cooling package: condenser, charge-air cooler and radiator; were represented by three separate porous regions where inertial and viscous resistance tensors were specified to have a realistic pressure drop and air flow through the cooling system. Two cooling fans were enclosed in separate polyhedral mesh regions where the rotation was simulated with MRF (multiple reference frame) method.

Rotating wheels were simulated using moving wall boundary condition with MRF regions between the spokes. Even though it is known that the tyre tread may affect the airflow and forces distribution [12], in order to make simulation less computationally expensive and the results more reproducible and easier to compare, it was decided to use slick tyres. As previously mentioned the tyres were morphed to simulate the actual shape of loaded tyre and contact patch.

The numerical wind tunnel was created with dimensions large enough not to result in any blockage or interference effects.

The boundary conditions for the tunnel were set as follows:

pressure outlet, moving ground for the floor with the same velocity as the inlet, and symmetry plane boundary conditions for the sidewalls and the ceiling of the virtual wind tunnel.

The final mesh size depending on the case had 75-90 million volumetric cells.

Simulations were carried out using steady-state approach in RANS (Reynolds-Averaged Navier-Stokes) mode with the k-epsilon realizable turbulence model using coupled solver. The cases were considered to be converged when the residuals were stable bellow 10^{-3} and both drag and lift coefficients (C_D and C_L) variations were lower than ± 0.5 drag counts for the last 500 iterations.

Cases Investigated

Firstly, in order to investigate the ventilation torque acting on the rotating wheels and its dependencies, two different wheel rim designs were used, see [Figure 4](#).

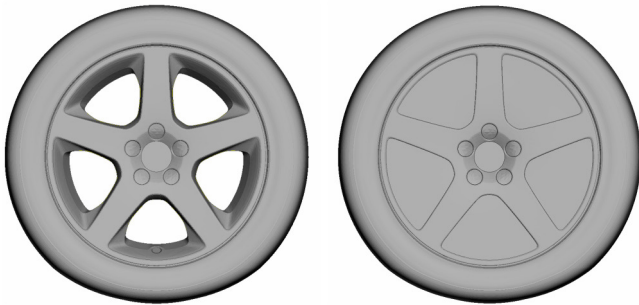


Figure 4. Two wheel rim designs investigated

The fully covered rim is known for decreasing aerodynamic drag of the vehicle [13] and producing relatively low ventilation torque [9], [8]. The first four cases investigated included Baseline configuration with standard rims, case with just front wheels covered, case with just rear wheels covered and the case when all rims were changed to fully covering rim. These four cases were investigated at a vehicle velocity of 100 km/h.

Secondly, the dependency of ventilation torque on the vehicle velocity was investigated. For that purpose the model with two different rims configurations (Baseline and All Covered) was tested under changing wind speed conditions with a corresponding adjustment made to MRF regions and moving wall boundary conditions. The six velocities tested were in the range from 60 to 160 km/h.

Equivalent Force, Ventilation C_D

In order to compare different cases to each other it is convenient to introduce the force equivalent to the ventilation torque, F_{vent} , which is going to be called ventilation resistance force in this study. This force may be introduced in the following way using power requirement to overcome ventilation torque:

$$F_{vent} \cdot V_{vehicle} = P_{vent} = \sum_{i=1}^4 T_{vent,i} \cdot \omega_i \quad (1)$$

where $V_{vehicle}$ is vehicle velocity, P_{vent} is a total power required to overcome ventilation torques $T_{vent,i}$ acting on every wheel; ω_i is the rotational velocity of the wheels. Keeping in mind that

$$V_{vehicle} = \omega_i \cdot r_i' \quad (2)$$

where r_i' is deformed wheel radius, that changes with the speed and can be different for front and rear axle, the [equation \(1\)](#) can be rewritten in the following form:

$$F_{vent} = \sum_{i=1}^4 \frac{T_{vent,i}}{r_i'} \quad (3)$$

To be able to compare contributions of aerodynamic resistance force and ventilation torque it is also possible to write ventilation resistance force F_{vent} in the same way as usual notation for aerodynamic drag force:

$$F_{vent} = \frac{1}{2} C_{D(vent)} \rho A V_{vehicle}^2 \quad (4)$$

where A is a reference frontal area of the vehicle and ρ is air density. $C_{D(vent)}$ is being called ventilation resistance coefficient. Using [equations \(3\)](#) and [\(4\)](#), $C_{D(vent)}$ can be calculated from the ventilation torque using the following formula:

$$C_{D(vent)} = \frac{2}{\rho A V_{vehicle}^2} \cdot \sum_{i=1}^4 \frac{T_{vent,i}}{r_i'} \quad (5)$$

This formula can be simplified assuming that the deformed radius of the wheels r_i' is the same for all four wheels:

$$C_{D(vent)} = \frac{2 T_{vent}}{\rho A V_{vehicle}^2 r_i'} \quad (6)$$

Here T_{vent} is a sum of ventilation torques of four wheels, or the ventilation torque of the vehicle.

Defining $C_{D(vent)}$ in this way makes it non-dimensional and allows summarizing this coefficient with aerodynamic drag coefficient $C_{D(aero)}$ in order to be able to address total aerodynamic resistance of the vehicle, see [equation \(6\)](#):

$$C_{D(aero+vent)} = C_{D(aero)} + C_{D(vent)} \quad (7)$$

As seen from the definition this coefficient includes both effects of ventilation torque and aerodynamic drag and hence allows better comparison of different vehicle designs to each other.