

A Method for Truck Underbody Aerodynamic Investigation

Richard G. Stephens, Patrick R.R.J. Stevens, and Holger Babinsky
University of Cambridge

ABSTRACT

The underbody of a truck is responsible for an appreciable portion of the vehicle's aerodynamic drag, and thus its fuel consumption. A better understanding of the underbody aerodynamics could lead to designs that are more environmentally friendly. Unfortunately there are difficulties with correctly replicating the ground condition and rotating wheels when using the classical approach of a wind-tunnel for aerodynamic investigation. This in turn leads to computational modelling problems. A lack of experimental data for Computational Fluid Dynamics (CFD) validation means that the flow field in this area has seldom been investigated. There is thus very little information available for the optimisation and design of underbody aerodynamic devices. This paper investigates the use of a water-towing tank, which allows the establishment of the correct near-ground flow while permitting good optical access. Using a 1/10 scale model, Reynolds Numbers of around 0.7 million are achieved. A novel double light-sheet planar PIV setup is used to measure the flow field. This arrangement of two coplanar horizontal laser sheets provides data in regions that would typically be zones of no information with a single laser sheet arrangement. Two high-speed cameras are used, and their images are stitched together to give a large field of view at high resolution. It is found that this new experimental arrangement allows detailed quantitative flow data to be collected throughout the vehicle underbody. These results may then be used to provide a basis for the optimisation of underbody components.

CITATION: Stephens, R., Stevens, P., and Babinsky, H., "A Method for Truck Underbody Aerodynamic Investigation," *SAE Int. J. Commer. Veh.* 9(2):2016, doi:10.4271/2016-01-9020.

INTRODUCTION

The aerodynamics of commercial vehicles is becoming an increasingly pertinent topic. In 2012, Heavy Goods Vehicles (HGVs) in the UK consumed nearly 8 billion litres of fuel [1]. High fuel costs along with heightened pressure to protect the environment mean that commercial operators are looking for ways to reduce vehicle fuel consumption.

When travelling at constant velocity on flat terrain, the fuel consumption of a vehicle is directly proportional to the resistive force that it must overcome. The total resistive force can be simplistically broken down into a rolling resistance and an aerodynamic drag component. [Figure 1](#) shows how the force components vary with vehicle velocity, based on the simplified force relation given by [Equation 1](#).

$$F_D = \frac{1}{2} \rho_a U_\infty^2 C_D A + N C_{rr} \quad (1)$$

Values for the $C_D A$ (drag coefficient multiplied by the frontal area) and C_{rr} (rolling resistance coefficient) are from coast-down testing undertaken by the Centre for Sustainable Road Freight on a European truck [2], and are assumed to remain constant with velocity (in reality they vary slightly with velocity). At European motorway speeds of 56mph, the aerodynamic drag is accountable for approximately 45% of the power requirement. There is thus scope to make a significant reduction to the fuel consumption by lowering the aerodynamic drag.

The aerodynamic drag of commercial vehicles is frequently broken down into four sources, the cab, the gap between the cab and the trailer (the cab gap), the rear, and the underbody. Of particular interest, due to its lack of previous consideration, is the underbody. Approximately 15-30% of the aerodynamic drag is attributed to this area [3,4].

The majority of published literature concerning the underbody of a HGV focuses on the performance of specific aerodynamic devices. The most popular device is the side-skirt, which has been shown to be effective by numerous studies [3, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14]. However, there is little direction for the design of underbody flow devices as very few investigations consider the fundamental physics of the underbody flow.

One reason for the lack of existing studies may be the unsuitability of traditional testing methods. The underbody flow is close to the ground, and is affected by the establishment of a correct ground condition, as well as the rotation of the wheels. Significant differences have been observed in wake topologies by Fritz and Ortega et al. [15, 16] and in wheel flows by McManus & Zhang [17] when an incorrect ground condition is used. The requirement of a moving ground plane and rotating wheels is seldom achieved in a wind-tunnel, requiring an expensive rolling road set-up which hampers optical access. The complexity of the underbody geometry coupled with a lack of experimental data for validation also introduces problems for Computational Fluid Dynamics (CFD) modelling. Traditional methods of investigating HGV aerodynamics

are thus not suitable for the underbody, and another method is required. Furthermore, Cooper's 1984 study indicates that it is important to correctly represent the underbody detail [18]. He found that models that are too simplistic can alter the drag of the vehicle by up to 10%.

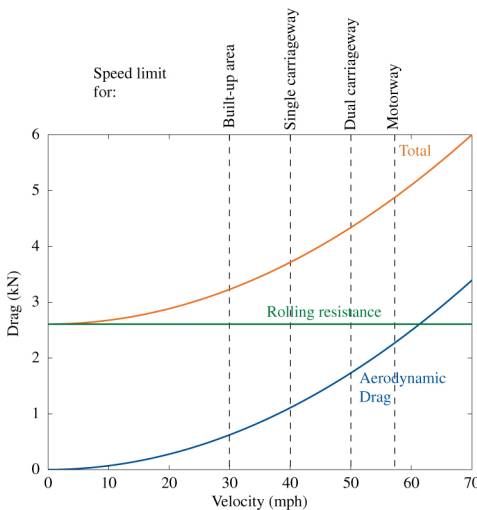


Figure 1. Aerodynamic and rolling drags as functions of the vehicle velocity.

For an experimental campaign to provide a useful contribution, the approach should thus be capable of the following: (i) replicating the ground condition and rotating wheels, (ii) capturing the complex geometry of the vehicle underbody and (iii) giving a quantitative and qualitative picture of the flow in order to understand how improvements can be made.

METHODOLOGY

Water Towing Tank

The experiments make use of the Cambridge University Engineering Department's water towing tank facility, shown in Figure 2. Whilst similar facilities have been used for other automotive applications (for example, by Morgan [19]), a water towing tank has not previously been used for studies on HGVs.

The water towing tank is 7m long with a usable cross-section of 1×0.8 m. It has a belt-driven translating carriage. With no load, the carriage is capable of reaching speeds up to 4.5m/s. A typical maximum loaded speed is around 3m/s. The side walls and floor of the working section of the tank are made from clear acrylic, allowing good optical access.

Between each experimental run, the tank must be allowed to settle for an adequate amount of time to allow turbulent fluctuations to die down. In the case of the results presented in this paper, at least 20 minutes are left between each run. Morgan [19] determined that for this facility, turbulent fluctuations are reduced to a free-stream level of 0.02% after 20 minutes.

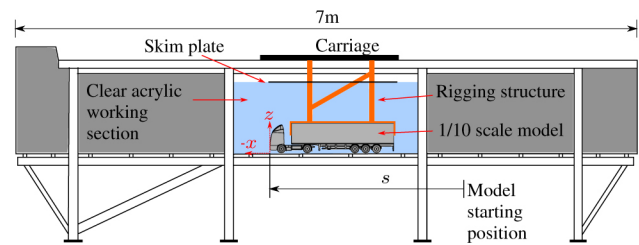


Figure 2. Schematic drawing of water towing tank.

The model is rigged to the moveable carriage with the wheels touching the ground using a vertical mounting structure. The ride height of the vehicle, taking into account the compression of the vehicle wheels, is measured using a camera set-up. The height is constant for all measurements to an accuracy of ± 0.75 mm. A horizontal plate spanning the width of the tank and the length of the truck is fixed to the carriage. The plate skims the top of the water, maintaining a constant water height and pressure head. It serves an additional purpose of damping surface waves that occur following an experimental run.

The Model

To fulfil the requirement to realistically capture the complex underbody geometry, a new model was designed. Figure 3 shows a schematic of the model. Its dimensions are equal to the 1/10th scale equivalents of the UK road-legal limits [20]. Its geometric features have been defined from an analysis of several different manufacturers' designs, in order to create a 'generic' vehicle. It is designed to be watertight to minimise its mass and thus reduce the force required to accelerate it. Great care was taken to correctly represent the underbody detail. In particular, the model includes the structural beams that form the chassis. It also has reconfigurable components such as: a fuel tank, air cylinders, axle diagonal cross bracing, side barriers, side-skirts, and rear bumper. Additionally, the model is equipped with functional wheels which rotate when the model is towed. This paper presents results from a basic configuration as shown in Figure 3. This configuration comprises: a wheel and axle assembly, a bumper, a secondary fuel tank located between the rear of the tractor and the trailer wheels, landing legs, and two air tanks.

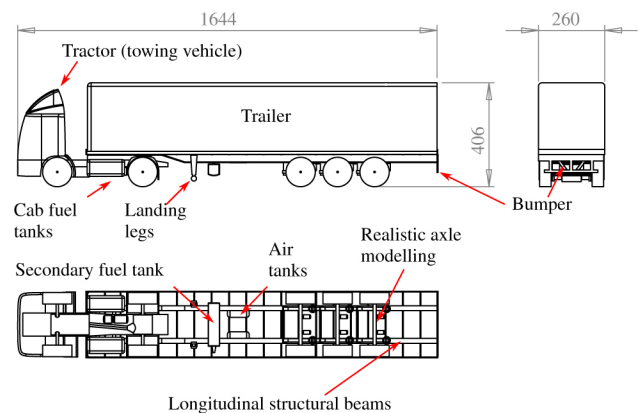


Figure 3. Scale drawing of truck model (dims in mm).

The model in the working section of the towing tank creates a blockage of 17%. Although this is relatively high, its most significant effect would be on the absolute drag forces. An application of Maskell's method [21] leads to an approximate drag coefficient correction of $C_D = 0.7C_{D, \text{measured}}$. However, this is not expected to greatly affect the underbody flow as the effect of blockage on the static pressure distribution around the vehicle has been found to be small in similar investigations [22].

Velocity Profile and Dynamic Similarity

The velocity of model translation is measured using an opto-reflective sensor attached to the carriage reading a band of black and white stripes, spanning the length of the tank. Each stripe is 4mm wide, and the data is sampled at 24kHz. Each instantaneous velocity measurement is therefore accurate to $\pm 0.048\text{m/s}$, or $\pm 1.6\%$ of 3m/s . Figure 4 shows a typical velocity profile, where U_∞ is the desired free-stream velocity, u_i is the instantaneous velocity of the model and s/b is the normalised stream-wise position of the tractor leading edge in the tank. The velocity reaches the desired free-stream value just before $s/b = 5$. There is an initial overshoot of 8%, but this is accounted for in the normalisation process.

The measurement region, or the Field of View (FOV) of the PIV cameras, is located between $s/b = 5.4$ and $s/b = 6.3$ (where s/b gives the normalised distance from tractor leading edge before an experimental run). As the model has a width-normalised length of 6.3, it remains in the FOV for the majority of the constant velocity region and begins to slow when the rear of the trailer is approximately one normalised length downstream of the FOV. Due to the time taken for viscous effects to develop, the effect of its deceleration on the flow field will be delayed. However, it is expected that flow in the wake of the vehicle more than two widths downstream of the trailer rear will be affected by unsteady effects due to this deceleration.

Due to a slightly varying model velocity and the final deceleration, the velocity used for normalisation is non-constant. Instead, results presented in this paper are normalised by the velocity u_{norm} according to:

$$u_{\text{norm}} = \begin{cases} u_i & \text{if model within FOV} \\ U_\infty & \text{if model outside FOV} \end{cases} \quad (2)$$

Water temperature is measured in order to correctly determine the density and viscosity needed to calculate the Reynolds Number. The maximum speed of translation of the model is around 3m/s . This corresponds to a width-based Reynolds Number in the range of $Re_w = 6.3 \times 10^5 - 7.8 \times 10^5$, depending on the water temperature at the time measurements are taken. This range is of the order of the minimum value recommended by SAE guidelines for truck aerodynamic testing ($Re_w = 7 \times 10^5$) [23].

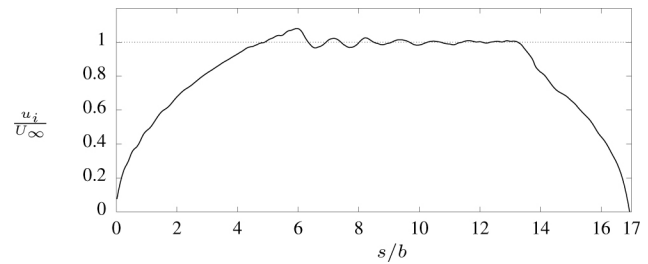


Figure 4. Typical velocity profile.

High Speed PIV System

Particle Image Velocimetry (PIV) is used to obtain flow velocity information. The system comprises a double cavity high speed laser with a maximum frequency of 20kHz, capable of producing an output of 30mJ per pulse per cavity at 1kHz at a wavelength of 527nm. Two Phantom Miro 310 high speed cameras are used to obtain planar data, with a maximum resolution of $1280 \times 800\text{px}$. Particles of titanium dioxide are used for seeding the flow. They have a characteristic diameter of $45\mu\text{m}$, and are approximately neutrally buoyant. Images are checked for peak locking by ensuring that particles are smeared over several pixels, and that the probability density function of the processed velocity field shows no tendency towards integer velocities.

This paper presents results from a 2D planar set-up with a pair of laser sheets formed horizontally from the side of the tank, at varying z positions. A schematic diagram of the setup is shown in Figure 5. The locations of the laser sheets with respect to the model used in this investigation are shown in Figure 9, which takes account of tyre compression. The laser sheets are aligned to the measurement plane with an accuracy of $\pm 1\text{mm}$. To reduce the area of laser shadow regions formed by underbody components, measurements are taken with a laser sheet emanating from both sides of the tank following the method of Stevens [24]. This is achieved using a beam splitting optic and a series of mirrors which direct the beam into a secondary sheet forming optic on the other side of the tank facility. To give a larger field of view, two cameras are used and their images are overlapped and stitched together. They are arranged side by side underneath the tank, with their image axes normal to the measurement plane. The acquisition frequency in this case is set such that a velocity of 3m/s corresponds to a particle movement of 5 pixels: 2.5kHz. Prior to the stitching process outlined below, the field of view of each camera is approximately $340\text{mm} \times 214\text{mm}$. PIV analysis windows of $16 \times 16\text{px}$ are subsequently used, giving a vector resolution of 4.3mm .

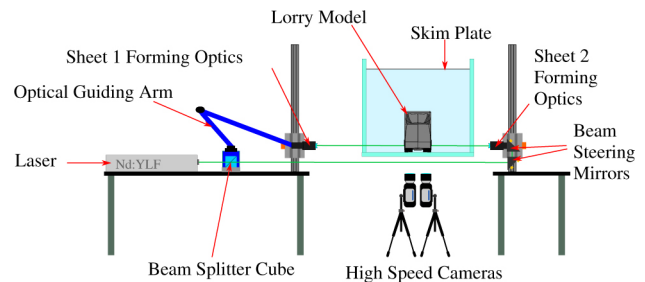


Figure 5. Planar PIV Setup, end view.