

Development of a Wind Tunnel Test Section for Evaluation of Heavy Vehicle Aerodynamic Drag at a scale of 1:3

Damien J. McArthur, David Burton, Mark Thompson, and John Sheridan
Monash University

ABSTRACT

Full scale heavy vehicle aerodynamic testing requires a very large wind tunnel test section, with few wind tunnels having this capacity worldwide. Small scale testing often requires a loss of model detail as well as introducing Reynolds Number and compressibility effects. A $\frac{3}{4}$ open jet wind tunnel set-up has been developed at Monash University Wind Tunnel that enables testing of 1:3 scale truck-trailer models, of full-scale length up to 18 metres to be tested.

The measured drag on longer vehicles is more strongly affected by horizontal buoyancy and long models create additional blockage when yawed. In addition the length of the model means that special care must be taken to ensure that shear layers emanating from the nozzle at the start of the test section are sufficiently separated from the shear layers and wake at the base of the truck.

In preparation for 1:3 scale testing the wind tunnel test section was lengthened, the jet size was increased and a false floor was added to reduce the size of the ground boundary layer. This paper details the modifications made and the following validation study undertaken. Presented are streamwise static pressure profiles, boundary layer evolution profiles, measurements of the growth of the jet shear layer and flow mapping planes to establish flow uniformity.

CITATION: McArthur, D., Burton, D., Thompson, M., and Sheridan, J., "Development of a Wind Tunnel Test Section for Evaluation of Heavy Vehicle Aerodynamic Drag at a scale of 1:3," *SAE Int. J. Commer. Veh.* 6(2):2013, doi: 10.4271/2013-01-2455.

INTRODUCTION

Heavy vehicle fuel consumption is a significant problem in many countries. It is becoming increasingly important to reduce the amount of emissions produced. In 2010 Heavy vehicles in Australia consumed a total of 6,403 million liters of fuel with the average fuel consumption of articulated trucks being 56.2 litres per 100 kilometres [1]. As fuel costs increase the value proposition of fuel saving initiatives such as add-on aerodynamic devices is improving.

Significant progress has already been made in developing solutions that reduce the aerodynamic drag of heavy vehicles as reviewed by Cooper [2]. There still remains plenty of work to be done both to increase the uptake of these proven aerodynamic devices such as boat-tails, gap seals, side skirts, etc. Active flow control devices are immature and remain at a level where they are not being implemented in on road situations.

In Australia a significant portion of interstate freight is transported by trucks. Distances between the major cities are large, for example the major routes of Melbourne to Sydney,

and Melbourne to Brisbane, being approximately 900 and 1700kms respectively. Heavy vehicle highway speed limits are relatively high, typically of the order of 100km/h. Together, these large distances and relatively high speeds, create a significant potential for fuel consumption savings by aerodynamic optimization. It also creates a need for facilities and capabilities to locally evaluate and optimize heavy vehicle aerodynamics.

A challenge to the uptake of drag reduction devices on the road has been the lack of large scale wind tunnel testing at representative Reynolds numbers. The majority of experimental development of heavy vehicle aerodynamic technology has been completed at a scale of around 1:10, often on simplified bodies [3,4,5,6]. At such a small scale it becomes difficult to accurately represent detailed geometry and very high wind speeds are required in order to replicate on road conditions, this can push the power limits of many wind tunnels and lead to issues with compressibility. A notable exception to this is the work done at NRC by Leuschen and Cooper[7] who tested drag reduction add-on devices on a full size truck-trailer combination at a blockage

ratio of 13.3%, a feat achievable at only a few of the larger operational wind tunnels. Testing has also been completed at full scale at NASA AMES [8] and at representative Reynolds Number at scale in pressurized wind tunnels [9].

In an attempt to achieve a balance between Reynolds number, model scale and tunnel blockage issues a 1:3 scale heavy vehicle testing program has been initiated at the Monash University 1.4MW wind tunnel.

The models tested in this facility include the GTS reference model [10,11] and a detailed representation of a current generation cab-over-engine truck coupled to a detailed van type trailer (figure 1), the results of which are presented in [12]. These geometries were chosen as they are representative of common vehicle set-ups observed in the Australian transport industry. This paper details the modifications made to the Monash University 1.4MW wind tunnel in preparation for 1:3 scale heavy vehicle testing. Also presented are the results from the flow validation process.



Figure 1. 1:3 scale model in the Monash University 1.4MW wind tunnel.

WIND TUNNEL SETUP

The Monash University 1.4MW wind tunnel is a closed circuit type tunnel with a 3/4 open jet automotive test section. The tunnel is powered by 2, 0.7MW fans. Between the fans and the jet exit, the flow is contracted twice, firstly by a two sided horizontal contraction and then a one sided vertical contraction.

In standard configuration the jet is 4m wide and 2.6m high giving a jet area of 10.4m² and a total contraction ratio of 5:1. The test section is 9m long from the jet exit to the beginning of the collector. The origin of the coordinate system for the tunnel is the centre of the turntable, with the z axis pointing vertically and the x axis in the downstream direction (figure 2). Previous testing on a 1:3 scale GTS model showed that this configuration was unsuitable for testing a model this large as the shear layer emanating off the

roof of the jet impinges on the shear layer at the back of the truck.

A number of modifications were made to the tunnel to improve test conditions for heavy vehicle scale testing. The angle of the vertical contraction was reduced, raising the height of the jet exit to 3m, combined with a 0.27m high false floor for a total jet area of 10.9m² and a contraction ratio of 4.8:1. This had the effect of raising the shear layer off the top of the jet out of the vicinity of the model as well as reducing the solid blockage from 11.2% to 10.6%. This comes at the expense of a slightly reduced top speed, increased free stream turbulence and decrease in flow uniformity. As well as being raised, the jet exit was moved upstream by 0.9m. This increased the spacing between the nose of the model and the jet exit, minimizing nozzle blockage effects. In order to reduce negative pitch in the flow the vertical contraction was moved a further 2m upstream, with the final section of the jet being of constant cross section.

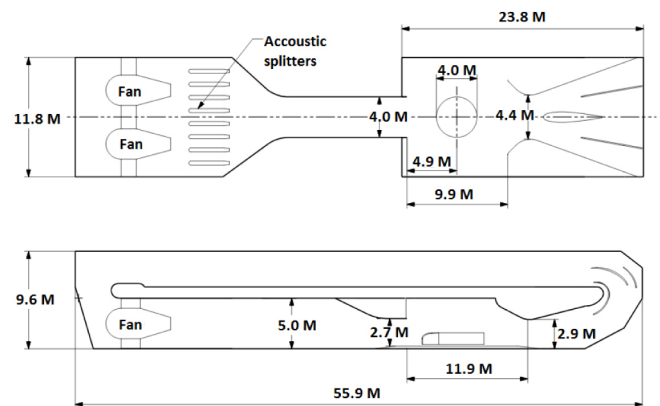


Figure 2. Monash University 1.4MW wind tunnel.

FORCE MEASUREMENT

Force and moment are determined using 4 Kistler type 3-component force transducers mounted in the wind tunnel floor. Supports extend down between the trailer wheels and are mounted to the force balance beneath the false floor (figure 3). The force balance is designed such that 3 components of force (drag, side and lift forces) and 3 moments (pitch, roll and yaw) are determined. The force balance is mounted to the turntable, so that at yaw angles the forces are measured in the model axis and no transformation is necessary.



Figure 3. Truck model mounted to force balance beneath the false floor.

INSTRUMENTATION

Flow field measurements were taken with a 4 hole dynamic pressure probe (cobra type) [13]. The cobra probe is capable of obtaining three components of velocity (u, v, w), static pressure, pitch, yaw, turbulence intensities (I_{uvw} , I_{uu} , I_{vv} , I_{ww}) and Reynolds stresses. The probe can only make accurate readings for flow at incidence angles of less than 45° and is unsuitable for measuring reversed flow. The precision of the velocity outputs from the cobra probe is 0.1m/s . Flow mapping was done at a test section velocity of 100km/h , corresponding to the maximum rated velocity for the probe traverse. Samples were taken at a rate of 500 Hz for 15 seconds.

All pressure measurements, including those from pitot static tubes placed in the test section were taken using a Dynamic Pressure Measurement System (DPMS) from Turbulent Flow Instrumentation. The DPMS has 64 available diaphragm channels, all connected to a common reference which was placed in a plenum outside the flow of the test section. The maximum range of the system is $\pm 3.0\text{kPa}$. A transfer function is incorporated into the data acquisition system to remove the effect on the transient response due to the length and diameter of the pressure tapping tube used.

FLOW VALIDATION RESULTS

Reference Dynamic Pressure

The wind tunnel is instrumented with a permanent array of 6 pitot static tubes located upstream of the test section between the two contractions. Previous testing has shown that the reference pitots are located far enough upstream to be

unaffected by test section blockage. In order to obtain the reference dynamic pressure in the test section, a relationship was found between the dynamic pressure at the upstream pitots (averaged across the 6 pitot static tubes) and the dynamic pressure at a reference position in the test section. This relationship differs based on the configuration of the test section and contractions, therefore a unique dynamic pressure factor must be found each time the tunnel geometry is modified. Test section dynamic pressure was measured using a pitot static tube located above the centre of the turntable at a height of $z = 450\text{mm}$. The dynamic pressure ratio between the upstream pitots and the test section was found to be $q_1/q_0 = 0.787$. The maximum test section velocity achieved was 47m/s corresponding to a width based Reynolds number of 2.4×10^6 for the 1:3 scale models tested.

Static Pressure Gradient

For long models such as heavy vehicles the empty tunnel, streamwise static pressure gradient can have a significant effect on the measured drag coefficient. The static pressure gradient through the test section was investigated via two methods, both by pressure tapping the test section floor and by placing pitot static tubes at various streamwise locations.

Floor static pressure measurements

A central row of pressure taps was located in the floor of the test section along a line 50mm offset from the centre of the tunnel. Other taps were located at $y = \pm 415\text{mm}$ from the centre line of the tunnel, corresponding to the location of the sides of a typical 1:3 scale heavy vehicle model. This allowed for static pressure variations to be averaged over the width of a model. In all there were 20 taps per row in the streamwise direction at a nominal spacing of 0.5 to 1m . All tests were run for 60 seconds, 3 times each at a speed of 100km/h .

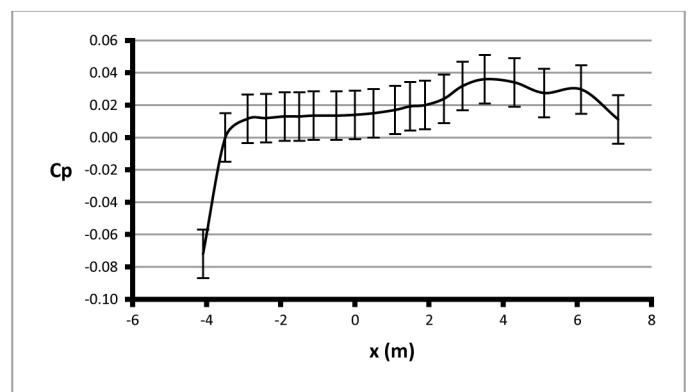


Figure 4. Static pressure gradient at floor pressure taps.

The results from the floor pressure taps are shown in [figure 4](#). A steep rise in C_p can be seen from $x = -4\text{m}$ to $x = -3.5\text{m}$, followed by a more moderate increase until around $x = 6\text{m}$ before reducing again. There is a large amount of scatter in the data due to the boundary layer and the effects of any local surface imperfections. The mean static pressure