

Evaluation of the Influence of Stakes on Drag and Fuel Consumption for a Tractor-Logging Trailer Combination

Mithun Shetty and Marius-Dorin Surcel
FPInnovations

ABSTRACT

The main objective of this study is to reduce the aerodynamic drag of tractor-trailer combinations used in the forest industry. In most cases, logging trucks on their return trips are usually travelling in unloaded conditions with upright stakes, which add drag. CFD and wind tunnel testing suggested a drag reduction of up to 35% with no upright stakes, which corresponds to 17% in fuel savings in unloaded conditions. One of the proposed fuel reduction concepts was therefore to have foldable stakes so that the stakes could fold down into a horizontal position while travelling in unloaded conditions.

Fuel savings of 15% for a vehicle with stakes in the horizontal position were confirmed with track testing when compared to the fuel consumption of a vehicle with stakes in the vertical position. The coastdown test indicated 28% reduction in drag. The difference in drag reduction between the coastdown test and initial simulation was due to stake size and profile. The simulation was therefore conducted again with the stakes' profile similar to the one used in the track test and the same drag reduction of 28% was obtained.

The foldable stakes concept can cut annual fuel consumption by 3000 to 4000 liters, with GHG emissions reduction of up to 8 tonnes per vehicle. Trailer manufacturers should look into the possibility of incorporating foldable stakes into hayrack semi-trailer configurations in order to maximize potential fuel savings, with safety taken into account in the design process, for vehicles operating under certain conditions.

CITATION: Shetty, M. and Surcel, M., "Evaluation of the Influence of Stakes on Drag and Fuel Consumption for a Tractor-Logging Trailer Combination," *SAE Int. J. Commer. Veh.* 7(2):2014, doi:10.4271/2014-01-2447.

INTRODUCTION

Transportation is a huge cost for the forest industry, representing up to 50% of the delivered cost of raw fiber. In Canada, roughly 2.7 billion liters of fuel are consumed per year, which equates to 7.3 million metric tonnes of greenhouse gas (GHG). With haul distances expected to increase in the future, transportation activities will likely have an even greater impact on the competitiveness of the Canadian forest industry and will also significantly affect the life cycle analysis of forestry [1]. In addition, rising fuel prices and a goal to minimize the industry's carbon footprint are making forest companies look at ways to reduce their consumption of diesel fuel during logging operations and during the delivery of fiber to mills.

Forest companies require a program of targeted fuel studies to identify operating techniques and technologies that can reduce their fuel consumption and/or increase their productivity. This project primarily aims to reduce energy costs by focusing on improving trailer aerodynamics. It has been shown that aerodynamic improvements can increase fuel economy by 11.5% in a tractor-trailer combination [2].

Although aerodynamics account for close to 50% of the power requirements at high speeds, aerodynamic designs have rarely been considered for forestry vehicles. In the past, relatively low fuel prices meant that the anticipated benefits for forestry applications were not sufficient to pursue. However, with the price of fuel increasing, this is no longer necessarily true. Furthermore, new and innovative composite materials allow a great degree of flexibility in creating new vehicle shapes. New futuristic truck cabs are just an example of this.

Currently, aerodynamic improvements for such trailer configurations as logging trucks, chip trucks, flatbeds, etc. are not being fully developed and more research is required in this area.

The primary objective of this study is to introduce new drag reduction technologies for logging trailers. One way to minimize the drag would be to fold the stakes when there is no load on the truck, as it is quite evident that the stakes in a vertical position add unnecessary drag when a truck returns from the mill to the loading site. To quantify the drag reduction

computer simulations were conducted and correlated with wind tunnel tests, coast down tests and track fuel consumption test results.

METHODOLOGY

Both computational and experimental methods were used as investigative tools to develop the next generation of aerodynamic trailers. Wind tunnel data was used to compare the CFD model results [3]. Fully validated CFD analyses allowed rapid evaluations of a number of potential technologies, allowing research to continue on the most promising ones. The CFD analysis identified folding stakes as the most promising, and subsequent research was concentrated on this strategy.

Wind Tunnel

Wind tunnel test results were used to correlate the CFD results. The experiments were conducted in a closed-loop wind tunnel characterized by a quasi-circular test section of 0.927 m in diameter and 1.83 m long (Figure 1). The model was mounted on an elevated ground board that was installed in the middle of the test section in order to minimize the effect of a build-up of thick boundary layers on the tunnel walls. The total length of the ground board was 1.248 m and the front-wheel axis of the truck model was located at 0.211 m downstream from the leading edge. The ground board therefore extended about two body widths upstream from the model, as per SAE J1252 standard [4]. The thickness of the ground board was 0.016 m and the ground board extended over the entire width of the test section [5]. The small boundary-layer was also built up between the wheel and the ground floor, and therefore a clearance of 8.7 mm was provided between the wheels and the ground floor. The displacement thickness over the ground plate was estimated to be in the range of 2.4 to 7 mm depending on the nature of flow. A 1:24 scaled model of a Kenworth T800 tractor with a logging semi-trailer was used in the tests. The forces were measured with an Aerolab LLC pyramidal force/moment balance.

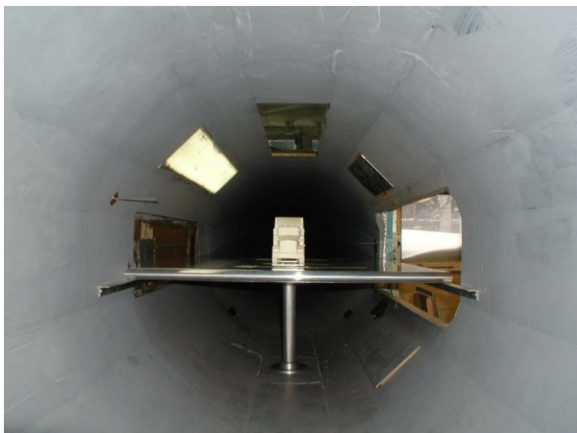


Figure 1. Wind Tunnel

The maximum wind speed with the truck model installed in the test section was about 28.3 m/s (102 km/h) and the temperature was set to 22.7 °C. A Reynolds number (Re) of 1.86×10^5 was achieved using a model width of 0.1 m and kinematic viscosity of air under standard pressure conditions. This Re value was lower than the Reynolds number stipulated by the SAE J1252 recommended practice [4], which is $Re_{SAE} = 7 \times 10^5$. The objective of the recommended practice is to ensure the comparability of aerodynamic data coming from different wind tunnels. In the present study, even if $Re < Re_{SAE}$, the flow is considered to behave like a high Reynolds number flow where inertia forces are dominating. In this case, if separation points are at fixed positions on the surface of the body, the drag coefficient is considered to be independent of Re . The Reynolds number is therefore not high enough to allow comparison with other experiments in large wind tunnels, but it is high enough to build a significant data base for the purpose of CFD validations [5]. The variation of aerodynamic coefficients with the Reynolds number is illustrated in Figure 2. For the range of the Reynolds number being tested, the aerodynamic coefficients didn't vary much for the Reynolds number of 90,000 and higher.

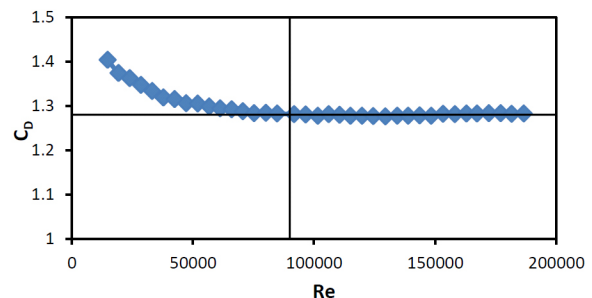


Figure 2. Variation of drag coefficient at a zero degree yaw angle with different Reynolds numbers in wind tunnel

A tractor wheelbase of 0.23 m (5.6 m in full-scale) was maintained throughout the test. The SAE J1252 wind tunnel test procedure was followed [4]. The wind-averaged drag coefficient (C_{dw}) was computed based on five yaw angles: -2.7° , 0° , 2.7° , 7.245° and 9° . The test was conducted with a fixed floor and static wheels. Since Canada's average wind is comparable to US average wind, the same methodology for computing C_{dw} was followed [6]. All the flow parameters were corrected for the effects of solid and wake blockage of the 3D model and for the effect of horizontal buoyancy induced by the growing boundary layers developed on the test-section walls [5].

CFD

For this study, Ansys CFD-Flo commercially-available software and Sharc's Harpoon Meshing software were used. The problem was solved on a Dell Precision T3500 Workstation that had 12 GB of RAM (1333MHz, DDR3 SDRAM, ECC-3 DIMMS) on a Quad Core Intel processor (Xeon® W3565 3.20GHz, 8M L3, 4.8GT/s) and a 64 bit Windows® Windows 7 operating system. A hex dominated mesh was used, and the domain size is illustrated in Figure 3, while Figure 4 shows the

mesh size close to the body and the boundary layer size. Fifteen layers of boundary layer with a boundary layer size of 0.0008 m were created around the truck to capture the boundary layer flow separation with Y^+ value around 5 or lower [7]. The computational mesh sensitivity was performed and the valid mesh size was used for all subsequent analyses. Table 1 presents the input set-up parameters for CFD models.

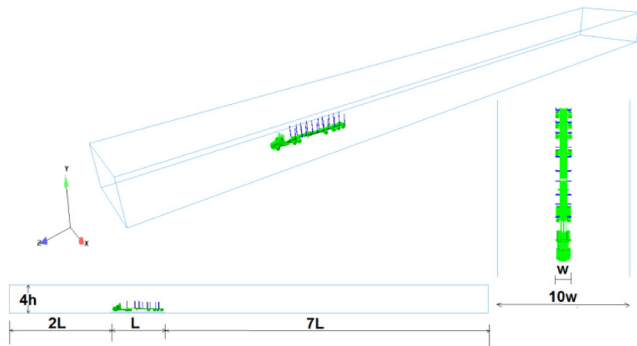


Figure 3. Domain size

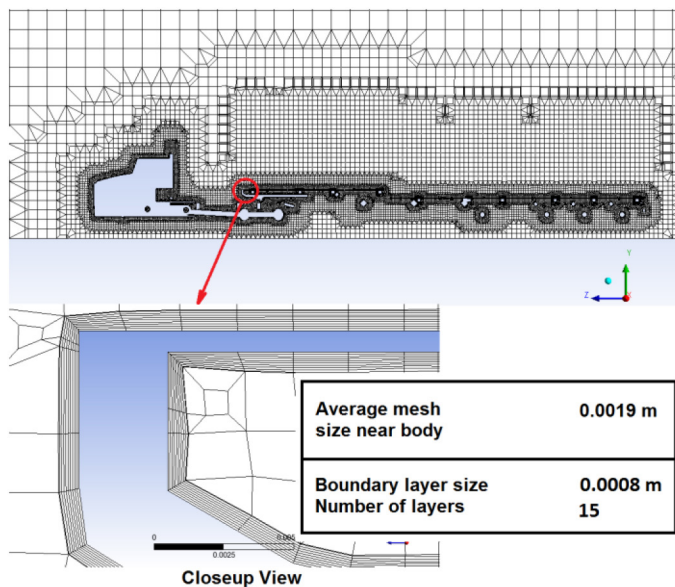


Figure 4. Mesh type and size

Table 1. Input parameter

Parameter	Type/ value
Analysis type	Steady state
Turbulence model	Shear Stress Transport
Turbulence intensity	0.5%
Eddy length scale	1 mm
Velocity	28.33 m/s
Density	1.19 kg/m ³
Advection and numeric scheme	High resolution

Table 2 depicts the boundary conditions applied to the CFD models. The turbulence intensity (TI) was less than 0.5% for the wind tunnel where all the tests were conducted; therefore, the CFD model was equally run at TI of 0.5%.

To simulate the wind tunnel condition, the tunnel floor was fixed and inlet conditions were applied at the entrance to the test section of the wind tunnel. Inlet velocity was set to the uniform velocity which was used in the wind tunnel test, of 28.3 m/s, and the relative pressure at the outlet was set to zero. A “no-slip” boundary condition was applied to the vehicle body, which means that the velocity at the wall is explicitly set to zero. A free slip condition was applied to the tunnel ground to avoid boundary layer formation. Initially, the ground plate was also modeled and a no-slip boundary condition was applied to the plate section; however, the drag coefficient results remained unaffected with this inclusion of the ground plate. The ground plate was therefore not modeled for the subsequent analyses.

Since a truck is subjected to cross-winds in the real world, and in order to obtain a drag value that corresponded to the average different cross-wind conditions, the C_{dw} was calculated based on the SAE J1252 procedure to correlate CFD results with wind tunnel results. The C_{dw} took into account the variation of drag coefficient with the yaw angle, the average speed, the wind direction relative to the vehicle and the vehicle speed.

Ansys CFX provides flexibility in choosing discretization schemes for each governing equation. The discretized equations, along with the initial condition and boundary conditions, were solved using the high resolution solution method. A first-order high resolution discretization scheme was used for the momentum equation and turbulent quantities, where high accuracy is required in the presence of shocks or discontinuities. In general, these schemes ensured satisfactory accuracy, stability and convergence. For many applications, steady state solutions are preferable, and for many applications a good Reynolds Average Navier Stokes (RANS) model with a good quality grid will provide all the required accuracy [8]. The steady state RANS CFD method was therefore used for all runs in this study. The convergence criterion for both the continuity and momentum equations was set to 10^{-5} , which meant the residuals were very small. However, the criterion was not strictly enforced as the focus was to have the drag coefficient parameter converge and a residual value for the momentum equations in the range of 10^{-3} and 10^{-4} , which were considered acceptable in this study. To make sure that the solution was stabilized, the convergence was checked from the residuals and the drag plot. The iteration step was varied from 100 to 200 steps, depending upon each case, in order to obtain reasonable accuracy in predicting drag. The time step was set to 0.05 sec for all the runs. Ansys CFX-Solver uses this time step as a means of under-relaxing the equations as they iterate towards the final solution [8].

Table 2. Boundary condition

Boundary type	Boundary details
Inlet, Left tunnel wall, Right tunnel wall	$U = \text{Input Velocity} * \sin(\theta * \text{Yaw Angle}/180)$ $V = 0$ $W = -\text{Input Velocity} * \cos(\theta * \text{Yaw Angle}/180)$
Outlet	Pressure: 0 Pa
Tunnel roof and ground	Free slip condition
Vehicle body	No slip

Fuel Consumption Track Test

The track test was performed to quantify the fuel savings of the most promising concept. For the vehicle testing, the main activity was controlled testing on a track at Transport Canada's Motor Vehicle Test Centre (MVTC) [9] in Blainville, Quebec, which is presently operated by PMG Technologies. The fuel consumption tests were performed on the high-speed test track (BRAVO). The track is a high-banked, parabolic oval with a length of 6.4 km (4 miles). The length of a test run was 87 km (approximately 13 laps). Figure 5 presents the test site.

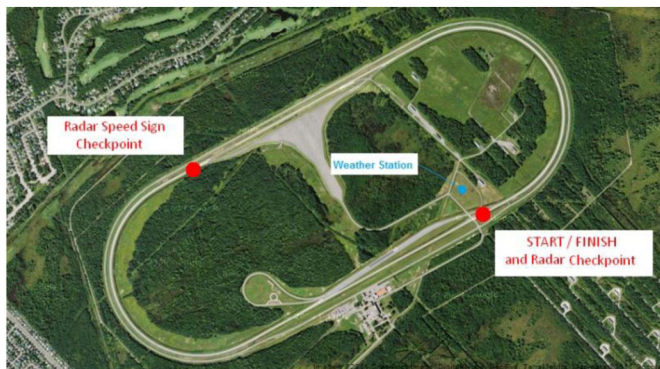


Figure 5. Map of test track with radar checkpoints and weather station location.

The test and control trucks were both 2007 Kenworth T800 with extended cabs. The control truck was coupled to a shortwood (also known as a hayrack) style logging trailer with 20 stakes; the test truck was coupled to a similar logging trailer with 20 stakes. The vehicle specifications of the tractors and trailers used for the tests are listed in Table 3. Figure 6 presents the test vehicle and the control vehicle. The vehicles were in good working condition, with all settings adjusted to the manufacturer's specifications, and confirmed to be in proper operating condition prior to and during the tests.

The test vehicle was modified by removing all the stakes and strapping them on top of the trailer (Figure 7) so that the weight of the test vehicle remained the same during both runs, i.e. baseline and final runs.

Table 3. Vehicle Specifications

	Control	Test
	Tractors	
Make and model	Kenworth T800, tandem drive	
Date of manufacture	03/2007	03/2007
Engine make and model	Cummins ISX-500	Cummins ISX-500
Rated power	500 HP	500 HP
Transmission	RTLO16918B	RTLO16918B
Differential ratio	4.3	4.3
Tires	Michelin XZA-1, XZA-3 11R22.5	Michelin XZA-1, XZA-3 11R22.5
Tire pressure (cold) psi	100	100
Tare weight (kg)	9960	9910
Average tread depth	Steer 12 mm (15/32 in.); Drive 18 mm (23/32 in.)	Steer 11 mm (14/32 in.); Drive 19 mm (24/32 in.)
Initial odometer reading (km)	319621	318394
	Trailers	
Type	Quad with lift axle	
No of axles	4	
Tires	Michelin 385/65 R22.5; 11R22.5 XZY3	Michelin 385/65 R22.5; 11R22.5 XZY3
Tire pressure	100	100
Tare weight (kg)	10425	8630
Average tread depth	10 mm (13/32 in.)	10 mm (13/32 in.)



a). Control truck



b). Test truck with stakes