

# Aerodynamics of an Unloaded Timber Truck - A CFD Investigation

Petter Ekman and Roland Gardhagen  
Linköping University

Torbjörn Virdung  
ANSYS Sweden

Matts Karlsson  
Linköping University

## ABSTRACT

Reducing energy consumption and emissions are ongoing challenges for the transport sector. The increased number of goods transports emphasize these challenges even more, as greenhouse gas emissions from these vehicles increased by 20 % between 1990 and 2013, in Sweden. One special case of goods transports is the transport of timber. Today in Sweden, around 2000 timber trucks transport around six billion ton kilometers every year. For every ton kilometer these vehicles use around 0.025 liter diesel, and there should exist large possibilities to reduce the fuel consumption and the emissions for these vehicles. Timber trucks spend most of their operation time travelling in speeds of around 80 km/h. At this speed aerodynamic drag contributes to around 30 % of the total vehicle resistance, which makes the aerodynamic drag a significant part of the energy consumption. One of the big challenges with timber trucks is that they travel unloaded half of the time. This put higher demands on possible drag reduction modifications, as they need to function and be practical for both when the timber truck is loaded and unloaded. In this study an unloaded timber truck has been investigated by use of computational fluid dynamics. The recently released Stress Blended Eddy Simulation model has been used for simulating the flow over a timber truck at a Reynolds number of 1.1 million, based on the square root of its frontal area. From the results it could be seen that 52.8 % of the drag is generated by the cab. By investigating a drag reduction device that covered the gap between the bulkhead and the first stake pair, a drag reduction up to 6.7 % was possible, which shows potential for simple modifications that not influence the daily usage.

**CITATION:** Ekman, P., Gardhagen, R., Virdung, T., and Karlsson, M., "Aerodynamics of an Unloaded Timber Truck - A CFD Investigation," *SAE Int. J. Commer. Veh.* 9(2):2016, doi:10.4271/2016-01-8022.

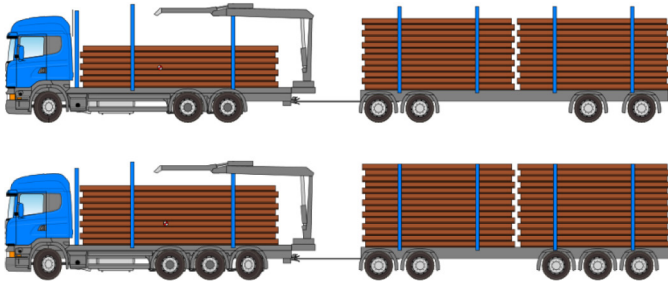
## INTRODUCTION

The demand for reduced emissions and energy consumption are ongoing challenges for the transport sector. Between 1990 and 2013 a 20 % increase of greenhouse gas emissions from road goods transports, has been seen in Sweden, as a result of increased amount of goods transportation [1]. A significant cause of the energy consumption and hence the source for emissions, is the aerodynamic drag. Typically, goods transportation trucks travel in speeds of 80 km/h, where the aerodynamic drag is responsible for 25 to 50 % of the energy consumption, depending on the shape and weight of the vehicle [2], making aerodynamics an area of interest for potential reduction of energy usage.

One special case of transports is the transport of timber. In 2015, about 2000 timber trucks in Sweden, hauled about six billion ton-km (metric ton kilometers) of timber [3]. These timber trucks have an averaged fuel consumption of about 0.025 liter diesel per ton-km, which is between 30 to 50 % higher than ordinary tractor-trailer configurations.

No encouraging improvements have been seen for the averaged fuel consumption over the last 20 years for these timber trucks. One of the main reasons for this is the poor aerodynamic development of these vehicles. Typically, between 20 to 30 % of the fuel consumption is caused by the aerodynamic drag for a fully loaded timber truck [4], and the number become even higher when the truck is unloaded. Even though a lot of aerodynamic development has been done on the cab, rarely the entire vehicle has got the proper attention. There exist several reasons for this, where one is that the truck manufacturers' development focus is on tractor for tractor-trailer configurations, as it constitutes most of their customers. Another reason is the number of manufacturers involved in the making of the complete timber truck. Typically, the cab and chassis of a timber truck are manufactured by a truck manufacturer, while several other manufacturers are responsible for the stakes, bunks, trailer, bulkhead, and for some configurations a truck mounted crane. Often these manufacturers (except the truck manufacturer) do not have the resources or knowledge of investigating their parts influence on the aerodynamic performance.

In Swedish forest industry there is currently a lot of effort put in different areas to reduce the fuel consumption and develop more environmentally friendly transport vehicles. One of the areas is to increase the maximum weight of the timber trucks from 60 tons to 74 tons, without increasing the length of the vehicles, [Figure 1](#). If designed properly this can lead to less number of required vehicles and also increase the freight efficiency of these vehicles [3].



[Figure 1](#). Top: Example of the most common 60-ton timber truck configuration with 1-2-2-2 axle configuration. Bottom: 74-ton timber truck configuration with 1-3-2-3 axle configuration.

The tractive resistance of a ground vehicle can be divided into four separate parts [2], [Equation 1](#).

$$F_t = F_{ROLL} + F_{ACC} + F_{CLIMB} + F_{DRAG} \quad (1)$$

where  $F_{ROLL}$  is the rolling-,  $F_{ACC}$  the acceleration-,  $F_{CLIMB}$  the hill climb- and  $F_{DRAG}$  the aerodynamic drag resistance.

For a vehicle driving with constant speed on a relatively flat road the rolling resistance and aerodynamic drag are the dominating forces. While the rolling resistance varies linearly with the speed of the vehicle, the aerodynamic drag varies with the square of this speed, [Equation 2](#), which increases the importance of it at higher speeds.

$$F_{DRAG} = \frac{1}{2} C_D \cdot \rho_{air} \cdot U_{\infty}^2 \cdot A \quad (2)$$

where  $C_D$  is the drag coefficient,  $\rho_{air}$  the density of air,  $U_{\infty}$  the free stream velocity and  $A$  the reference area.

Extensive studies of drag reduction devices exist for the tractor-trailer configuration, e.g. [2, 5, 6, 7, 8, 9, 10, 11] to name a few. However, for timber trucks only a few studies exist for its aerodynamics [3, 4, 12, 13, 14]. These studies also mostly cover class 8 tractors and how the loading of the timber affect the drag.

One substantial difference to regular truck transportation vehicles is that the timber truck travels unloaded half of the time. This does not only have an impact on the weight of the vehicle but also affects the shape of the vehicle, and hence the aerodynamic performance. From previous wind tunnel tests a 3.3 % higher wind averaged drag value has been seen for the unloaded timber truck compared to when it is fully loaded [3, 15]. At smaller yaw angles this increase could reach 10 %. In order for drag reduction devices to be feasible they need to

work for both when the timber truck is fully loaded and unloaded and also be structurally robust, as these vehicles operate in harsh terrain and weather conditions.

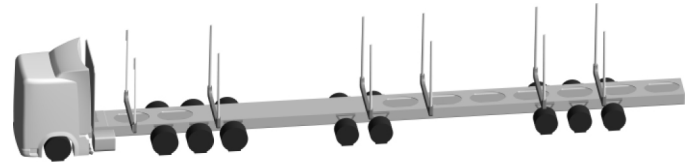
In this study the flow around an unloaded timber truck is investigated by use of computational fluid dynamics (CFD). The investigated timber truck is a geometrical updated version of the one studied in [3].

## AIM

The aim of this study is to investigate the flow around an unloaded timber truck, by use of CFD. A geometrical modification between the bulkhead and the first stake pair is also to be investigated, in order to reduce the aerodynamic drag of the vehicle. This modification does, however, need to be feasible for both an unloaded and fully loaded timber truck and not limit the daily usage. A secondary aim of this study is also to increase the knowledge about these type of modifications for upcoming wind tunnel test sessions and full scale road tests.

## METHOD

The geometry consists of a 1:6 scale model of a 74-ton timber truck with 1-3-2-3 axle configuration, [Figure 2](#). The used geometry is based on an updated version of the model used in [3] and has been slightly simplified, as nuts, screws and the balance have not been included in the model.



[Figure 2](#). Baseline configuration of the investigated 74-ton timber truck model.

The dimension and used reference area of the model can be seen in [Table 1](#).

[Table 1](#). Dimensions and reference frontal area of the investigated model.

|                        |                       |
|------------------------|-----------------------|
| Length                 | 4.25 m                |
| Height                 | 0.70 m                |
| Width                  | 0.42 m                |
| Reference Frontal Area | 0.2889 m <sup>2</sup> |

## Numerical set-up

As flows around bluff bodies are highly unsteady and time dependent a transient simulation method is often necessary to achieve sufficient accuracy for simulations. Therefore, in this investigation the recently released Stress Blended Eddy Simulation (SBES) model has been used. The SBES model is built upon the same approach as for the Detached Eddy Simulation (DES) approach, by using a Reynolds Averaged Navier-Stokes (RANS) model for the near wall flow and Large Eddy Simulation (LES) further away from the wall [16]. However, the SBES model uses a different shielding function and definition of the grid scale [16]. The shielding function is such

stronger than in ordinary DES and Delayed Detached Eddy Simulation (DDES) models and also has an improved transition between RANS and LES regions. This makes it possible to specify different model formulations of the RANS and LES regions. From previous work [17] it has been seen that the  $k-\omega$  Shear Stress Transport (SST) model [18] as RANS model and the dynamic Smagorinsky sub-grid model [19] is the preferred choice for bluff body simulations, and has therefore also been used in this study.

The used computational domain consisted of a rectangular cylinder with a length of 40.2 m, a width of 6.5 m and a height of 5.6 m, Figure 3. The model was placed 15 m (3.5 model lengths) from the inlet. The frontal area of the model covered less than 1 % of the cylinder cross section area to minimize possible blockage effects.

The inflow to the domain was modeled with a velocity inlet (with a prescribed uniform velocity profile with turbulent intensity and turbulent viscosity ratio of 1 % and 50, respectively) and the outflow with an outlet with zero static pressure. The walls for the domain were modeled with free-slip condition as they were deemed to be too far away to affect the flow around the model. The floor of the domain was modeled as a stationary ground but was divided into two separate zones, where the zone adjacent the model (dark gray colored in Figure 3) was modeled with no-slip condition and the zones farther away (light gray in Figure 3) was modeled with free-slip condition. This was done to imitate the same boundary layer thickness build up in front of the model, as in [3]. All the surfaces of the truck were modeled with no-slip conditions together with stationary wheels. A free-stream velocity of 30 m/s was used which resulted in a Reynolds number of 1.1 million, based on the square root of the models reference frontal area (Table 1).

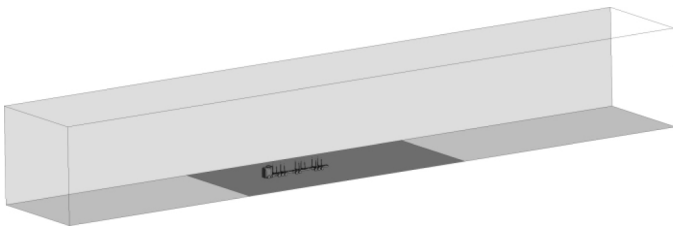


Figure 3. Overview of the used domain for the simulations. The dark gray regions are treated as walls with no-slip conditions while the light gray regions are treated as walls with free-slip condition.

All simulations were performed with the commercial Navier-Stokes solver ANSYS Fluent [20]. The Semi-Implicit Method for Pressure Linked Equations-Consistent (SIMPLEC) was used as  $p-v$  coupling scheme, least square cell based for the gradient discretization and the 2<sup>nd</sup> order discretization scheme for pressure [16]. The bounded central difference discretization scheme was used for the momentum equations, the 1<sup>st</sup> order upwind for the turbulent kinetic energy and turbulent dissipation rate, and the bounded second order implicit time integration was used for the temporal discretization [16].

A physical time step size of  $10^{-5}$  s was used to keep the Courant-Friedrichs-Lewy (CFL) number below 1 for essentially (>99.9 %) all cells in the domain. Five inner-loop iterations were used for every time step to ensure convergence below  $10^{-4}$  for the residuals. The simulations were initialized from a previous  $k-\omega$  SST RANS

simulation and run for a physical time of 1.0 s to remove possible effects from the initialization. Data was sampled during a physical time of 1.5 s and used for computing the time-averaged values. The sampled time corresponds to an air particle travelling 10.6 lengths of the model, based on the free-stream velocity.

The numerical set-up follows the SAE J2966 standard [21].

All simulations were performed on the Triolith cluster at National Supercomputer Centre at Linköping University, Sweden [22].

### Mesh Set-Up

The meshes were created in two steps. The surface meshes were created in ANSA [23], while the volume meshes were created in ANSYS Fluent [20]. The surface mesh consisted of triangle and quad elements with an edge length of 1.5 to 4.5 mm. Inflation layers were created on all the surfaces of the truck and on the ground, to properly capture the near wall flow and the boundary layer build up in front of the model. The inflation layers consisted of hexahedral and prisms elements. The first node height was set from 0.01 to 0.02 mm to ensure a  $y^+ \leq 1$  for the trucks surfaces. The first node height on the ground was set to give the first cell an aspect ratio of 12. This resulted in a varied  $y^+$  value of less than 5 adjacent to truck and between 30 and 100 in the rest of the domain. 10 to 20 inflation layers were created on the surfaces of the truck, while 10 layers were created on the ground. The volume mesh consisted of a Cartesian grid with refinement zones where high pressure and high velocity gradients were expected, Figure 4. The inflation layers and the Cartesian grid were connected by use of tetrahedral and pyramid elements. The mesh consisted of 56 million cells for the baseline configuration. A cell skewness quality of less than 0.95 was achieved for both the meshes.

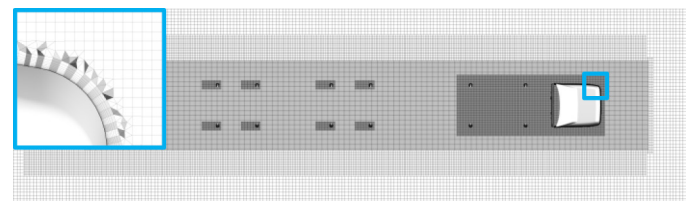


Figure 4. Volume mesh for the baseline configuration seen in a plane (at half the cab height) from above. Mesh refinements can be seen around the cab and the stakes. Inserted: Near wall mesh around the a-pillar of the cab.

### Modification

Seen in [15] the cab is responsible for most (57 %) of the drag for an unloaded timber truck, as it causes a large low pressure wake. By adding a device which increases the base pressure of the cab and the bulkhead, may therefore lead to a substantial drag reduction. Boat tails and cab extenders are typically add-on devices which are used for controlling the wake and increasing the base pressure of trucks [5, 8, 9, 10]. In [3] a boat tail behind the bulkhead showed promising potential by reducing the wind averaged drag value by 12 % for a fully loaded timber truck.

A similar concept was investigated in this study where a shield was placed between the bulkhead and the first bulkhead stake (Figure 5). The idea of the modification was to have a passive add-on device