

# Fuel Consumption Reduction by Geometry Variations on a Generic Tractor-Trailer Configuration

Antoine Devesa and Thomas Indinger  
FluiDyna GmbH

## ABSTRACT

Although considerable efforts have been made with respect to the reduction of fuel consumption of trucks during the last decades, the diminishing natural resources as well as the evolution of the truck traffic require continuous improvements in the field of aerodynamics. Indeed, the forces generated by the air on the trucks may originate, depending on weather, road type, truck type, dimension, etc, up to 50 % of the fuel consumption [1].

In order to analyze the influence of proportion variations (mainly related to the length) and add-on devices on the aerodynamic performance of a truck, a representative model was first generated. This simplified geometry of a tractor-trailer was based on the geometrical data of six European OEMs: Daimler, Iveco, and MAN (tractors), Kögel, Krone and Schmitz Cargobull (trailers). The model included a reduced level of details (exterior mirrors, wheels, simplified underbody and engine block).

The following processing chain was used for the numerical investigations. The grid generation tool, Spider [2-3], is an octree-based preprocessor producing boundary fitted, locally refined, conformal meshes, including hexahedral cell layers close to walls. The computational grid obtained was then used for the spatial discretization of the Reynolds-Averaged Navier-Stokes equations that are solved in parallel using the open-source software OpenFOAM® [4]. For comparison purposes the generic truck was built on a scale of 1:2.5 and investigated in Daimler's wind tunnel in Stuttgart [5] parallel to the numerical study. Both experimental and numerical results were found to match in terms of flow topology and drag coefficient.

Furthermore, an optimization analysis based on the generic body was carried out, where geometric variations or fittings for the front and the rear of the tractor-trailer were considered. The wind tunnel models of the trucks were made in modules to enable the experimental testing of almost all of the optimized configurations that were simulated. The numerical results showed very good agreement with the experimental data. In particular, the drag improvements matched satisfactorily. Consequently, this study will be extended with numerical investigations focusing on the 25.25 meter long truck combinations (European Eurocombis).

**CITATION:** Devesa, A. and Indinger, T., "Fuel Consumption Reduction by Geometry Variations on a Generic Tractor-Trailer Configuration," *SAE Int. J. Commer. Veh.* 5(1):2012, doi:10.4271/2012-01-0105.

## INTRODUCTION

The diminishing natural resources as well as the evolution of road traffic require continuous improvements to reduce fuel consumption and emissions of greenhouse gases. Significant efforts have been undertaken in the past decades to produce less polluting engines, to reduce the weight of the vehicles or to enhance their aerodynamic features. This is particularly true for the aerodynamics of trucks. Indeed, it is well known that the forces generated by the air on trucks may originate, depending on weather, road type, truck type, dimension, etc, up to 50% of the fuel consumption [6].

The truck industry in Europe is bound to various constraints coming from the fact that the permissible greenhouse emissions are regularly reduced and several laws restrict the design of the vehicles that are authorized to operate on the roads. For this reason, the evolution of truck

designs in an environment-friendly way is fairly restricted. Length, height and width are limited, and only some exceptions are allowed, which do not include aerodynamic devices. The objective of this study is to show the effect of aerodynamic devices or proportion changes on the aerodynamics and consequently on the fuel consumption, that would fit legal restrictions if they were slightly modified or if the permitted exceptions included aerodynamic devices. Since the infrastructure (bridges, streets) already constrains the height and width of vehicles, the focus of this study lies in the vehicle length variations.

An extensive literature research has been conducted in order to identify the most promising and realizable strategies for aerodynamic drag reduction and reduced fuel consumption. Only passive techniques were considered since they do not need any power or air supply that would reduce their efficiency.

The aerodynamic actions cannot be analyzed without taking into account the entire vehicle [7]. Indeed, if the tractor is already optimized for a certain trailer, the aerodynamic properties of the trailer will emerge more strongly and play a more important role. For this reason, it is mandatory that the flow around the tractor remains attached. The undercarriage flow should lose the least energy possible through underbody components. At the rear end, the flow should be modified such that the topology of the wake changes and the pressure on the rear of the vehicle increases.

## FRONT DESIGN

The lateral edges of the front of the vehicle play the most important role in the aerodynamic properties of the truck tractor. The lateral edges are responsible for detached flow regions where the kinetic energy of the flow is lost. In order to keep the flow attached, sharp edges are usually avoided and replaced by smooth, rounded edges. A critical radius  $r_{min}$  for the flow around these rounded areas can be estimated according to the incoming flow velocity and the viscosity of the air [1]:

$$r_{min} \approx 1.3 \times 10^5 \frac{\nu}{u_{\infty}} \quad (1)$$

When a radius greater than  $r_{min}$  is utilized, the flow is expected to stay attached. At an air speed of 80 km/h ( $\approx 22$  m/s),  $r_{min}$  is estimated at around 88 mm. Up to the present there has been no investigation exclusively dedicated to this topic of front radius variation, the manufacturer Scania published however a possible fuel consumption reduction of 1.2 % studying the drag changes on a soft nose configuration [8]. The concept of the soft nose is the forward extension of the front end of the truck in a smooth round manner, in the same manner as seen on the noses of high speed trains. Nevertheless, the truck model in that study was modeled without cooling air flow and with a total extension about 1 m.

It has been shown that other elements of the tractor play a significant role in the aerodynamics of the whole vehicle. These are for example the exterior mirrors, the lateral deflecting baffles, the front skirt, sun shields, etc. Most of their respective potential drag reductions have been analyzed in the past [1, 9] and gave a wide range of results (up to about 15 % drag reduction).

## ROOF AND LONGITUDINAL DESIGN

The spoiler of a truck exerts a great deal of influences on the flow on the upper part of the trailer, since it is meant to guide the flow to the trailer without separation. The benefit of the use of a spoiler depends on its type and settings. In the literature it can be found that the potential drag reduction is approximately 30 % [9]. For this reason, almost all trucks are equipped with such a spoiler, even though the setting is rarely optimized.

Several studies have been done in the past on the partial or complete covering of the undercarriage of the entire truck [10,11,12]. Most variations have shown high efficiency;

however, these measures are fairly impractical in reality, because of the limited access to the mechanical components and the increased risk of deterioration of both truck and covering.

Impressive reductions of air drag were attained with so-called “tear-drop” configurations. In the thirties some streamlined trucks were produced, where the trailer showed interesting designs following the streamlines of the flow. Unfortunately, very sparse data concerning the aerodynamics of such trucks is available [13]. Don Bur [14] first extensively tested this concept of a convex curved roof and obtained a drag reduction of about 42 %. Indeed, the classical separation on the front part of the trailer is completely removed, since the transition from the tractor to the trailer remains smooth. Moreover the wake is reduced due to the inclination of the flow at the back-end of the trailer. Consequently the flow loses less energy in the rear part of the truck. However, due to the required over height of the tear-drop (higher than 4 m), such trailers are not permitted in Europe.

## REAR-END DESIGN

The main objective of modifications at the rear end of trucks is to obtain the greatest reduction in energy losses as well as the highest pressure on the back-end surface. If aerodynamic devices, such as plane or rounded flaps [1, 9], are used, the vortices that are forming at the trailer back edges are modified in such manner that they lead the flow to the back part of the truck. This results in an increase of the pressure on this surface, and consequently diminishes the overall pressure drag.

There is a wide variety of back flap systems (flat or rounded, inflatable or solid, full or hollow, etc) with widely differing drag reduction potentials. The most effective types can reach a drag reduction greater than 10 %. These benefits must be considered in light of the feasibility and the security issues of such add-on devices.

In order to analyze the influence of geometry variations (mostly related to the vehicle length) or add-on devices on the aerodynamic performance of a truck, a representative model was first generated. This generic geometry of a tractor-trailer is introduced in the Section “Generic Truck Configuration”. Then the numerical and experimental results of this study are presented in the following sections, including the analysis of aerodynamically relevant modifications at the front and back of the generic truck.

## GENERIC TRUCK CONFIGURATION CONCEPT

In order to gain insight into the aerodynamics of truck configurations, independently of individual OEMs designs, a generic truck was created. It was conceived to model the main features of existing vehicles, however, with a limited amount of detail. Each of the six European OEMs that participated in this project supplied a representative geometry

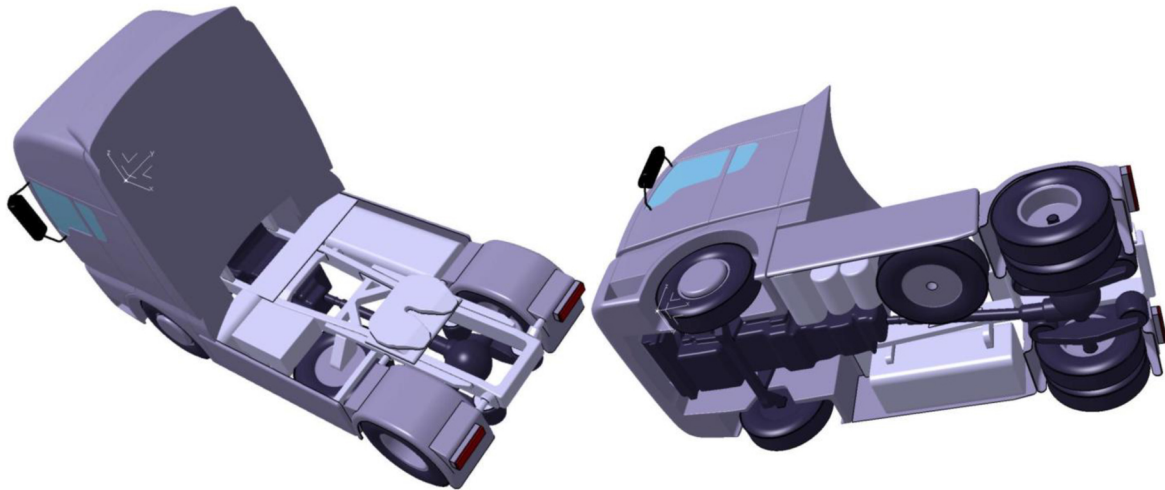


**Figure 1. Generic tractor model**

of either a tractor (Daimler, Iveco, MAN) or a trailer (Kögel, Krone, Schmitz Cargobull). Those geometries were “averaged” by means of a Computer Assisted Design (CAD) tool, resulting in a new representative and objective truck design.

The tractor was built by averaging the front and side surfaces of the available geometries, resulting in a truck body shell without any recognizable branding elements or features (Fig. 1). The spoiler was included in the roof geometry of the cabin at a relevant angle, allowing a smooth transition from the cabin roof to the front edge of a trailer with a height of 4m.

Various elements were not considered in order to reduce the level of detail of the generic geometry: sun shield, horns, third exterior mirror, side deflecting plates... Furthermore, several undercarriage components were excluded to simplify the geometry; only one spare wheel, fuel tank, transmission components... were included, since they play a dominant role in the underbody flow through their form and size.

The generic truck was equipped with a cooling air circuit for the engine, since it is known to significantly influence the flow around the driver cabin and the undercarriage. However, the modeling of a detailed engine compartment was avoided. The engine itself and the transmission were wrapped in a blockwise structure represented in Fig. 2. Instead of porous media, a simplified radiator and a condenser were designed with slots to produce the pressure loss of the cooling air before reaching the engine. The use of this type of construction is advantageous not only for the computational setup of the simulations, but also for the wind tunnel model, since no porosity has to be estimated.

Concerning the trailer, due to volume considerations, the load space remains is nearly constant for all manufacturers (of prismatic form) so that the maximum load can be used. The differences between the OEM trailer geometries are mostly concentrated on the undercarriage. The main chassis beam as well as the pattern of the cross members were simplified (Fig.3). All sidings were excluded, but a pallet compartment on one side was incorporated, as well as two

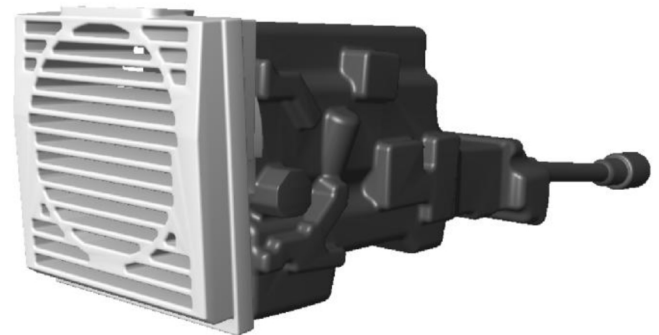


**Figure 2. Simplified cooling system and engine block (including transmission components)**

spare wheels at the rear of the trailer. The axle positions and support elements were averaged from the three available geometries. The top side edges were rounded with a radius of 25 mm. The front of the trailer was designed with chamfer of 45° on the sides to produce the most representative flow around it.

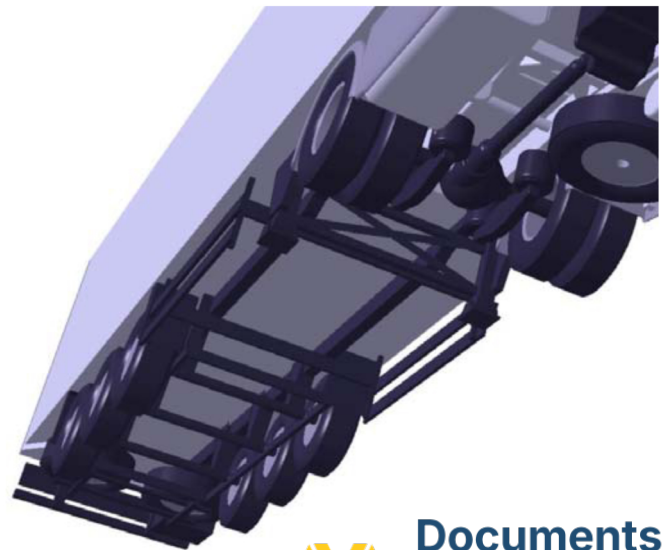


**Figure 3. Undercarriage of the generic trailer model**



**Documents**

SAE International

## WIND TUNNEL MODEL

In this study, not only numerical but also experimental investigations were undertaken. Since it is very difficult to experimentally investigate full size trucks, exceeding the dimensions of almost all wind tunnels, the generic truck geometry presented in the previous section was built on a scale of 1:2.5 (Fig.4).



**Figure 4. Complete wind tunnel model on a scale of 1:2.5 of the generic truck**

The main feature of the generic truck model is its modularity. The undercarriage components of the whole truck and the trailer itself were mostly made of metal, wood plates or plastic attached together rigidly. The tractor conversely was composed of a main block (Fig. 5) and exchangeable additional front and roof modules.



**Figure 5. Base module of the truck cabin**

## NUMERICAL AND EXPERIMENTAL METHODS

### GEOMETRY DESIGN / MORPHING

The geometry presented in a previous section was generated with the CAD software Catia V5. All the geometry variations that were analyzed during this study were also produced with Catia V5 or with the morphing tool PowerClay from the Exa Suite.

## COMPUTATIONAL SIMULATIONS

The computational grids for the numerical simulations in this study were produced with Spider [2-3]. Spider is an octree-based meshing method that produces boundary fitted, locally refined, conformal meshes with optional boundary layer cell rows in surface normal direction. It is massively parallel and automatically uses the available hardware without limitations. It is entirely parameterized, possesses a graphical interface but works as well in automated mode, which was a significant advantage for this project, where many configurations were investigated. The hexahedra dominant grids produced for this study were about 45 to 50 million cells. The smallest cell size was about 7 mm, which ensures that the first off-wall points were located between 50 and 180 normalized wall units ( $y^+$ ), which was small enough when using a high Reynolds number turbulence model with wall functions.

For the numerical simulations the open-source CFD software suite OpenFOAM® [4] was used. The suite contains many standard solvers that calculate the solution of Navier Stokes equations on a structured or unstructured grid for incompressible or compressible laminar or turbulent flows in or around complex geometries. In this study, the standard incompressible solver called simpleFoam was used. It solves the steady solution of the Reynolds-averaged Navier Stokes equations for turbulent flows. The use of unsteady methods like large-eddy simulations was not utilized, because they are more expensive in order to reach a time-converged state of the flow and average it then to obtain a global aerodynamic drag coefficient. The applied turbulence model was the SST variation of the  $k-\omega$  model of Menter [15], which use has significant advantages in the domain of external aerodynamics compared to  $k-\epsilon$  (size and location of detachment points...). All simulations were carried out at a speed of 22 m/s (80 km/h) with a straight inflow (no yaw angle). The ground had a slip velocity of 22 m/s. However for the sake of simplicity and increased computational performance, the wheels were not defined as rotating. The turbulent inflow and initial values were estimated such that the turbulence intensity was 1.8 % and the characteristic length of the turbulent scales was 10 mm.

The post processing of the simulations was made with the open-source software Paraview [16].

The specific feature of the numerical process chain was that all of the used software was open-source, with the exception of Spider. This allowed the process chain to be flexible and very low cost.

### WIND TUNNEL MEASUREMENT

All experimental investigations were carried out in the wind tunnel of Daimler AG [5], in Stuttgart, Germany. A six-point balance was used to measure the drag and several local and global flow pattern investigations were undertaken with the use of smoke lances (8- lance rack or single lance). The