

Heavy Truck Drag Reduction Obtained from Devices Installed on the Trailer

Luigi Salati, Federico Cheli, and Paolo Schito
Department of Mechanical Engineering, Politecnico di Milano

ABSTRACT

The flow-field around a “common” European heavy truck, equipped with several different trailer devices, is investigated using steady and unsteady simulations.

This work demonstrates how with simple devices added on the trailer it is possible to strongly decrease the aerodynamic drag over 10%, with an increase of overall dimensions below 1% without any change to the load capacity of the trailer.

Several devices, installed on the trailer, are tested on a target vehicle and the shape of the “airbag”, the “fin”, the “boat tail” and the “front-rear trailer device” has been optimized to achieve the maximum in drag reduction in front wind. The performance of the optimized devices are tested also in cross wind conditions with the yaw angle varying from 0° to 30°.

The truck equipped with the front-rear trailer device is also investigated using time variant simulation with yaw angle of 0°, 5°, 10°.

Detached eddy simulation (DES) with one equation Spalart-Allmaras as turbulence model is used to perform the analysis of the unsteady flow; the two equation $k-\omega$ SST turbulence model is selected for the steady-state RANS simulations. The unsteady simulations are performed using computational meshes on the order of 40 million of elements while RANS simulation are done with mesh of 10 million of elements.

The analysis is performed in the open-source CFD framework OpenFOAM®.

CITATION: Salati, L., Cheli, F., and Schito, P., "Heavy Truck Drag Reduction Obtained from Devices Installed on the Trailer," *SAE Int. J. Commer. Veh.* 8(2):2015, doi:10.4271/2015-01-2898.

INTRODUCTION

The research effort in heavy truck industry in recent years is focused on fuel efficiency.

The main motivations are:

- the regulations about emissions are increasingly more restrictive and require a continuous development on the final product;
- the rise of the fuel price;
- the optimization of the operational costs of the trucks.

It is estimated that 50% of the truck engine power is needed to overcome aerodynamic drag at typical highway speed of 90-100 km/h. In fact aerodynamic drag increases to the square of the vehicle speed, and is the main source of fuel consumption at cruise speed (Schoon [1]). It is estimated that a reduction of 20% in the drag coefficient results in 10% fuel efficiency increase (National Research Council Canada [2]).

A crucial point, for aerodynamic efficiency is the regulation concerning the maximum dimension for the truck-trailer. This is true especially in Europe (Directive 96/53/EC [3]), where the maximum permitted length for a heavy truck is measured considering the total length of the vehicle (tractor + trailer), while in the US the regulation prescribes only the maximum length of the trailer as shown in the [Figure 1](#). This regulation influences strongly also the tractor shape and its aerodynamic performance.

Many studies have been performed in the past decades both in wind tunnel and road test to investigate the flow-field around the truck-trailer. In the last years Computational Fluid Dynamics (CFD) studies are becoming more common due to the increase of High Performance Computing (HPC) performances and the computational power available for the simulation. The complexity of the geometry of the vehicle and the strongly-transient behavior of the flow-field require very detailed computational meshes: more than 20 million nodes are necessary to run a transient simulation with really small time step.

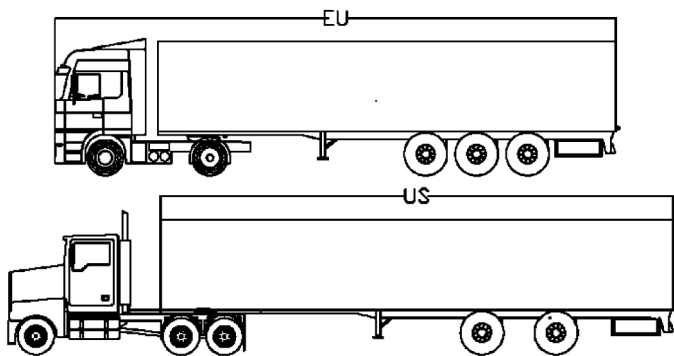


Figure 1. Comparison between European and American regulation regarding heavy truck maximum dimension.

The main areas that influence the overall drag coefficient of the heavy truck are:

- the front corner radius of the tractor (Palowski [4]);
- the tractor-trailer gap (Hucho [5]);
- the underbody of both trailer and tractor (Buchheim et al.[6], and Howell [7]);
- the interaction between the tractor and the trailer (Gilhaus[8]).

A wide range of devices can be developed for drag reduction on the tractor while for the trailer less solutions are practically used in the industry due to the limits of the current regulation and for technical issues (for example the access to crucial area of the trailer for maintenance, etc.).

Also economic reasons influence the current “box” shape of the trailer: box shaped trailers have higher load capacity compared to stream-line shape, easy access to the merchandise, easy delivery as a container on shipping boat, street dimension (height of the tunnel), etc. Another crucial aspect to take into account is the safety requirement of these devices: they must not increase the risk of injury for the passengers in case of crash with others vehicles.

The most common drag reducing devices are:

- cab roof fairings and side fairings (9-17% drag reduction - Drollinger [9]; Cooper [10]; Leuschen and Cooper [11]);
- trailer-front fairing (7-10% drag reduction - Garry [12]; Watkinset al. [13]);
- tractor and trailer side-skirts (4-6% drag reduction - Garry [14]; Ingram [15]);
- boat-tail and base flaps (7.5 % drag reduction - McCallen et al. [16]; Cooper [10]).

In recent years some studies have been performed on active drag reduction devices, such as boundary layer suction, but their complexity and the necessity to provide energy for their operation did not allowed to these devices to reach the market.

Another interesting way to reduce drag for the heavy truck transport is platooning as shown in Schito [17]. This solution requires a significant change in the infrastructure and in the driving behavior of the vehicle surrounding the platoon. For all these reasons commercial platooning is currently not available.

One of the main issues in the truck aerodynamics, is the clearance gap that has to be provided between tractor and trailer. If the gap is very small, the turning capacity of the vehicle will be strongly affected, while, with a large gap, the vehicle tends to behave as two separate bodies that generate twice the drag of a single body. This effect is amplified in cross-wind when the vehicle is yawed relative to the wind direction (Gilhaus [8]).

In this paper external devices that can be applied to the existing truck shape will be studied and developed in order to increase the aerodynamic efficiency of the truck, decreasing the fuel consumption of the vehicle with both economic and environmental benefit. The idea is to keep fixed the internal load capacity of the trailer and no radical changes on the truck body shape will be introduced: the proposed devices will be fixed to the existing trailer geometries. External devices can be easily applied to existing trucks with limited cost and high benefit in terms of fuels saving and installation costs. It must also be noticed that in this way no significant change must be done to logistic centers or shipping boats or in general to the whole commercial transport system. No modifications are proposed on the tractor since the modifications may influence also other features such as cooling, combustion efficiency and engine area arrangement.

The analysis will be conducted using open-source CFD software as OpenFOAM® and Paraview® [19], analyzing the flow around the truck using state-of-the-art solvers and methods.

TARGET VEHICLE

It is decided to use a truck geometry corresponding to a common truck currently available on the EU streets. The heavy-truck model is simplified for CFD studies (see Figure 2): the cooling system and bracket that holds the mirrors are neglected, as well as the turning lights and the vehicle registration plate holder.

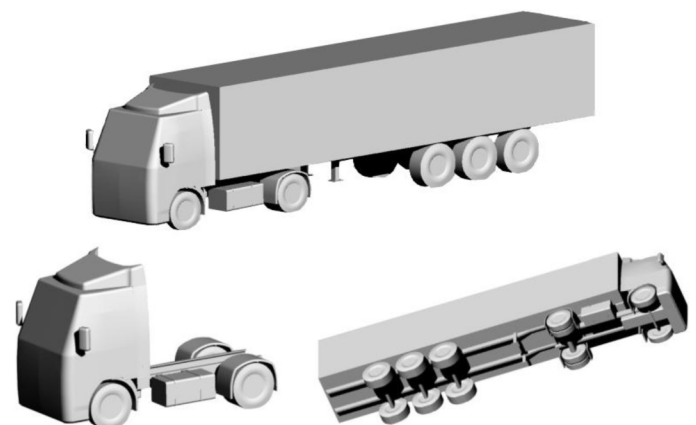


Figure 2. Heavy truck geometry simplified geometry used as target vehicle.

The base model is already provided with cab roof fairings, side fairings, wheel houses for the tractor tires and a suitable design of the front corner radius of the tractor. All these devices are currently available on most European heavy truck.

The lower part of the wheels are opportunely modelled to reproduce the contact area between the tires and the ground and the deformation of the wheels. The target vehicle overall dimensions (without the mirrors) are reported in [Table 1](#).

Table 1. Truck overall dimensions.

length	width	height
[m]	[m]	[m]
15.8	2.5	3.9

EVALUATED DEVICES

Taking into account only the aerodynamic features of the trailer, the main contribution to drag are: the rear of the trailer due to the vortices detaching from the top, the bottom and the sides of the rear faces of the trailer, and the underbody. This consideration leads to design aerodynamic devices especially influencing these areas. Different shapes are tested: “Airbag” (see [Figure 3-a](#)), “Fin” (see [Figure 3-b](#)), “Boat tail” (see [Figure 3-c](#)) and “Front trailer device” (see [Figure 3-d](#)). All the geometries are tested in both front and cross-wind.

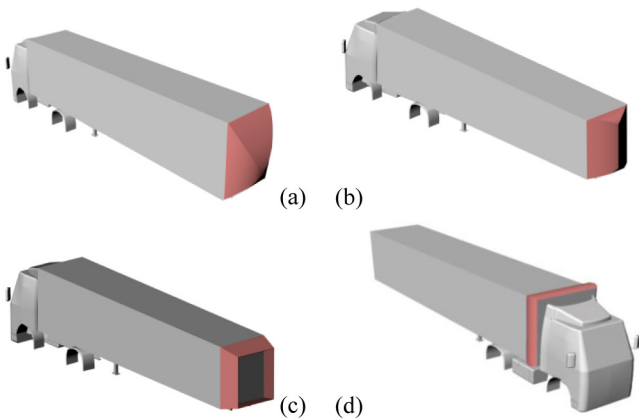


Figure 3. Devices installed on the trailer: Airbag geometry (a), fin geometry (b), boat tail (c), front trailer device (d).

The “Airbag” ([Figure 3-a](#)) is designed to decrease the impact on the drag of the vortices generated in the rear of the model, mainly from the top and the bottom of the trailer. It will work also as an airbag in case of crash when another vehicle is approaching one with this device installed in the rear part of it, if the structural design takes into account possible shock absorbing properties. A proper design for this device allows to access the trailer load area, but some issues may arise when placed on a cargo vehicle or in a cargo handling area since the space occupied by this device may interfere with the cargo infrastructure standards.

The “Fin” device ([Figure 3-b](#)) is modelled to influence the vortices generated in the rear of the model, especially from the sides of the trailer. Similar to the “Airbag”, also this device may have shock

absorbing features to be considered in the structural design phase. The “Boat-tail” devices ([Figure 3-c](#)) are designed to adjust the wake generated in the rear part of the trailer. They are currently installed on vehicles in the US, where larger maximum trailer sizes are allowed. The main issues with this device concern the stiffness of the boat-tail surfaces: in case of impact with a vehicle hitting another one with this device installed in the rear they may be dangerous as blades since they need to be realized using a resistant material. To ensure the access to the payload it is necessary that boat-tails are movable and to reduce the space requirements they may be foldable for parking purposes. The “Front trailer” device ([Figure 3-d](#)) is designed to move the separation on the front-side part of the trailer to a most suitable position located few centimeter above the surface of the trailer. It has a very small size and is positioned in a location that does not affects the access to payload and may have very low impact on the parking and storage of trailers.

When the boat tail or the fin or the airbag are installed on the trailer, the access to the payload is limited and some technical solution must be developed to have an access to the trailer inside without the need to uninstall the devices.

The increase of overall dimensions with respect to the standard truck-trailer configuration for each device is shown in [Table 2](#). Especially the increase of the width and height are an issue for the safety of the roads (as an example the height is connected with tunnels and bridges height and the lane size are fixed).

Table 2. Percentage of increase of the overall vehicle's dimension.

	% length	% width	% height
Airbag	5.1	0.0	0.0
Fin	4.8	0.0	0.0
Front trailer device	0.0	1.6	0.8
Boat tail	5.0	0.0	0.0

In order to prevent injuries in case of crash with others vehicle no sharp angle devices are tested (except the boat-tail) and all of them are designed to be constructed using soft materials.

NUMERICAL MODEL

The computational meshes are generated using the meshing tool provided by open-source CFD framework Open-FOAM®. The generated mesh is a fully Cartesian grid in the entire domain, with layers on the truck surface.

Different simulations are performed to optimize the computational domain dimension related to the physics of the phenomena especially in cross-wind. The computational domain (see [Figure 4](#)) is the optimum solution for both avoiding the blockage effect of the truck in the domain and decreasing the computational cost.

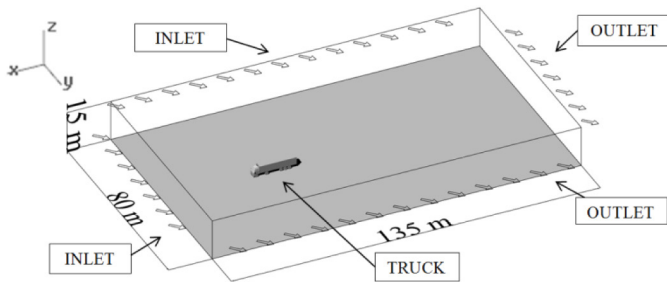


Figure 4. Computational domain and reference system.

The reference system is placed on the ground along the truck centerline. Positive X axis is directed according with the truck speed; positive Z axis is directed vertically and the Y axis has the truck transverse direction.

Two main rectangular volumetric controls, one inside the other are designed to refine the grid around the base geometry; other special refinement zones are defined to capture the wheel surface of the six tires. Several prismatic layers were added around the whole vehicle to correctly predict the flow in the boundary layer. Drag, side and lift force are computed as the force applied on the vehicle in x, y and z direction according to the reference system in Figure 4. Truck trailer aerodynamic force coefficients are calculated according to the following:

$$C_X = -\frac{F_X}{0,5 \cdot \rho \cdot U_\infty^2 \cdot A_{res}} \quad C_Y = \frac{F_Y}{0,5 \cdot \rho \cdot U_\infty^2 \cdot A_{res}} \quad (1)$$

$$C_Z = \frac{F_Z}{0,5 \cdot \rho \cdot U_\infty^2 \cdot A_{res}}$$

Where ρ is the air density, U_∞ is the speed of the air, A_{RES} is the frontal area of the vehicle for both front and crosswind.

Different grids from 10.8 million of elements to 48.5 million of elements are tested to check the grid independency on the target vehicle (see Table 3). The overall drag of the vehicle is used as parameter to monitor the grid independency in agreement with the SAE Standard J2966 [18]. The chosen grid, for the RANS simulations have around 10-11 million of elements (depending on the installed devices), the cells size around the truck is 0,015 m and the y+ value is around 30; while the unsteady simulations are run with mesh of 40 million of elements. In Figure 5 some mesh details are shown, assessing the good mesh quality and the reproduction of the prismatic cell layers all over the truck and trailer surface.

In Figure 6 the mesh detail of the tested devices are reported for the mesh of 10 million of elements.

Table 3. Grid independency study.

Million of elements	C_X	% C_X
10.8	0.586	
39.7	0.581	0.86
48.5	0.583	0.61

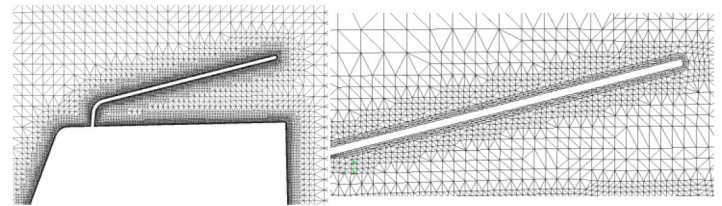


Figure 5. Mesh detail on the tractor top.

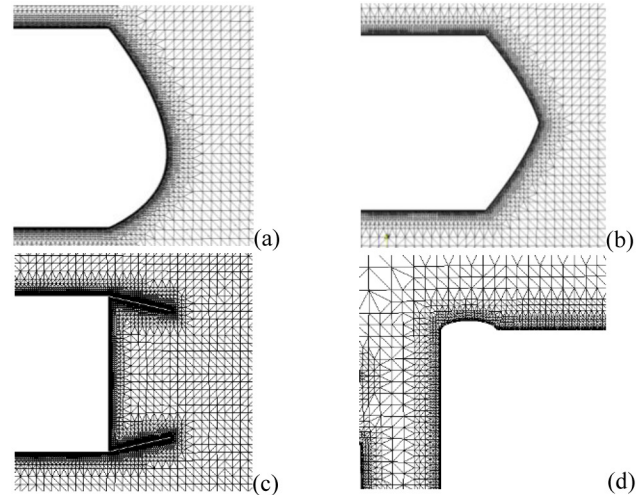


Figure 6. Mesh detail of the devices installed on the trailer: airbag geometry (a), fin geometry (b), boat tail (c), front trailer device (d)-

Time invariant incompressible RANS equations are solved using the simpleFoam [19] standard solver and the two equation k- ω SST turbulence model is used. Second-order upwind discretization scheme is used and SIMPLEC algorithm is adopted for coupling pressure and velocity [20].

The inlet velocity is set to 27 m/s; the ground has the same velocity of the inlet. This wind velocity generates a Reynolds Number higher than $7 \cdot 10^6$.

The truck is considered stationary while the flow and the ground have the relative velocity. The wheel of the truck and the trailer are put in rotation, in order to reproduce the operating condition of the truck. When analyzing only the devices performances, neglecting possible crosswind issues, the air speed (equal to the speed of the truck) is parallel to the x direction, zero pressure is set up at the outlet and slip wall is used for top and lateral boundaries. When cross wind effect are taken into account, the air velocity is setup in the x and y direction in both inlet and right side (where the x-component is the truck speed and the y-component is the crosswind velocity), while zero pressure is set-up in outlet and left side. Slip wall are used for the top. Yaw angle from 0° to 30° with and angle step of 5° are tested.

In order to speed up the simulation convergence and the computational time the steady state simulations are initialized with the potential solver potentialFoam [19]. RANS simulations are run until the convergence of the aerodynamic forces is reached.

When time invariant simulations are performed, Detached eddy simulation (DES) with the one equation Spalart-Allmaras [20] near wall model is used, transient equations are solved using the standard pimpleFoam solver [19].

DES simulations are performed using a time step suitable for the mesh size around the vehicle, paying attention to keep the Courant number below 1. The influence of different time steps were studied and the most suitable value for the time step is 10^{-4} s. Unsteady calculations were performed until statistically stable aerodynamic forces were calculated on the truck and the trailer.

Yaw angles of 0° , 5° and 10° are taken into account for the time variant simulations.

The set-up agrees with the SAE Standard J2966 [18].

FRONT WIND ANALYSIS

Target Vehicle

The Target vehicle has an overall drag of 0.586; the 14% of it is only generated from the wheels of the tractor and of the trailer.

From the normalized magnitude velocity contour plot in Figure 7, the region in which the main vortices are located is the rear of the trailer; the underbody part of it and the mirrors of the tractor. The vortices located in the rear of the trailer (see Figure 7 on the top) have different lengths: the top-rear vortex has a dimension almost double compared with the bottom-rear one, suggesting that it could be convenient to realize a device according with this non symmetric flow on the zero x-z plane. In Figure 7 on the bottom, the flow-field around the target vehicle is shown on the plane x-y at $z=2.6$ m where the mirror vortices and the side-rear ones are fully developed.

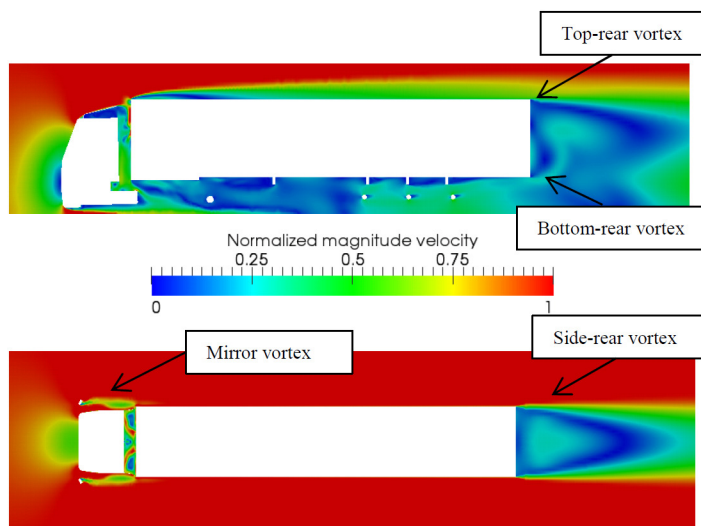


Figure 7. Normalized magnitude velocity contour plot. Top: vehicle centerline plane, bottom: horizontal plane at a height of 2.6 m from the ground.

Airbag

Airbags are designed to influence the rear of the truck vortices and are installed on the rear face of the truck; the airbag increases only the maximum length of the heavy truck (5% of the initial dimension). This rise in the overall length of the vehicle is not considered a big issue for the street safety. Especially if the device is made in soft material it will work also as an airbag in case of crash.

The main flow differences with respect to the target vehicle are shown in Figure 8: the bottom-rear vortex has a different shape compared to the one on the target vehicle, being largely reduced in length, leading to a drag reduction around 3.5%.

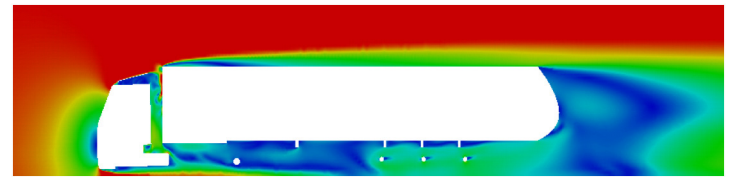


Figure 8. Normalized magnitude velocity contour plot at $y=0$ m.

Fin

As mentioned before a suitable design of the bottom part of the fin is crucial to obtain a high drag reduction from the device. This device increases the length of the heavy truck of 5% of the target vehicle length.

The fin shows high ability in reducing the overall heavy truck drag around the 8%. In this case the drag reduction is more evident than for the airbags, indicating that using a device that influences the lateral vortex structures it is possible to enhance the drag reduction.

The main advantage of this kind of device compared to the airbag one is in the reduction of the overall size of the vortices in the rear of the truck. In Figure 9, it is possible to see that the wake behind the truck is reduced with respect to the target vehicle one and justifies the drag reduction that is obtained.



Figure 9. Normalized magnitude velocity contour plot at $z=2.6$ m.

Front Trailer Device

This device is developed to minimize the impact of the device on the operation of the vehicle and to limit the costs of production compared to the previous devices and so promote their diffusion. It also guarantees an easy access to the payload, when the device is installed without the need to develop complex movement. Despite these benefits the front trailer device requires a minimum increase in the overall dimensions of the truck: about 1.6% increase of width and 0.8% increase in height, while no increase in the vehicle length is required. Both the raise in width and height may not be considered as an issue to the street safety. With a minimum increase of the overall dimension of