METHODS FOR THE DRAG REDUCTION
OF BLUFF BODIES AND THEIR APPLICATION
TO HEAVY ROAD-VEHICLES

G. Buresti, G.V. Iungo, G. Lombardi

DDIA 2007-6 October 2007

1st Interim Report
Contract between CRF and DIA
# List of contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 Introduction</td>
<td>2</td>
</tr>
<tr>
<td>2 Methods for bluff-body drag reduction</td>
<td>3</td>
</tr>
<tr>
<td>2.1 Generalities</td>
<td>3</td>
</tr>
<tr>
<td>2.2 Boat tailing</td>
<td>4</td>
</tr>
<tr>
<td>2.3 Base Bleed</td>
<td>8</td>
</tr>
<tr>
<td>2.4 Moving surfaces</td>
<td>10</td>
</tr>
<tr>
<td>2.5 Manipulation of boundary layer conditions</td>
<td>15</td>
</tr>
<tr>
<td>3 Further drag-reduction methods typical of tractor-trailer configurations</td>
<td>18</td>
</tr>
<tr>
<td>3.1 Tractor-trailer gap</td>
<td>18</td>
</tr>
<tr>
<td>3.2 Fairings and flow-deflection devices</td>
<td>19</td>
</tr>
<tr>
<td>3.3 Wheel-house shaping</td>
<td>20</td>
</tr>
<tr>
<td>4 Overview of the possible activities</td>
<td>21</td>
</tr>
<tr>
<td>5 Description of the chosen activities</td>
<td>24</td>
</tr>
<tr>
<td>5.1 Analysis of the aerodynamic features of a typical tractor-trailer</td>
<td>24</td>
</tr>
<tr>
<td>5.2 Development of devices comprising rotating cylinders</td>
<td>24</td>
</tr>
<tr>
<td>5.3 Manipulation of trailer boundary layer for base-drag reduction</td>
<td>26</td>
</tr>
<tr>
<td>5.4 Aerodynamic optimization of the mirrors</td>
<td>26</td>
</tr>
<tr>
<td>5.5 Control of tractor-trailer gap flow through blowing.</td>
<td>28</td>
</tr>
<tr>
<td>6 Temporal program of the activities</td>
<td>29</td>
</tr>
<tr>
<td>7 References</td>
<td>30</td>
</tr>
</tbody>
</table>
1 Introduction

The objective of the present report is twofold: reviewing the possible methods for the reduction of the drag of heavy road-vehicles, and devising a detailed research program to be carried out by the Department of Aerospace Engineering (DIA) of the University of Pisa within a research contract on this subject with Centro Ricerche Fiat (CRF). The activities envisaged by the contract are essentially computational, and limited to a time range of 9 months.

In the next section a brief review is given of some of the available methods for the reduction of the drag of bluff bodies in general, with particular attention to those that may be favourably applied in the design of heavy vehicles. Further drag-reduction methods typical of tractor-trailer configurations are then briefly described in section 3.

In section 4 the possible activities proposed by DIA are summarized, and in section 5 a more detailed description is given of those activities that were jointly agreed by DIA and CRF as the object of the present research contract. Their temporal development is finally reported in section 6.
2 Methods for bluff body drag reduction

2.1 Generalities

Bluff bodies are characterized by a more or less premature separation of the boundary layer from their surface, and by wakes having significant lateral dimensions and usually unsteady velocity fields.

Considering bluff bodies with fixed separation points, significantly different flow fields characterize the wakes of 2-D bodies perpendicular to the flow, and 3-D bodies that are elongated in the direction of the flow. Indeed, even if in both cases the wake velocity field is unsteady, the fluctuations are much higher in the first case, in which the wake flow is dominated by the alternate shedding of strong and highly concentrated vortices, producing the well-known Karman vortex street. The shedding of vortical structures is still present in the wake of axisymmetric bodies with a rounded leading edge (and thus with attached boundary layer up to the rear separation line), but these structures are weaker, less concentrated and less organized, and thus produce lower levels of velocity fluctuations. The latter are directly connected with the magnitude of the fluctuating forces acting on the body.

However, the above difference between the wake structures of 2-D and 3-D bodies has also a bearing on the connection between the flow field features and the mean aerodynamic forces, and in particular the mean drag. Indeed, highly concentrated vorticity structures imply the presence of a high perturbation energy in the wake, which is directly connected with both the instantaneous and the average drag. As a consequence, it is well established that the most effective way of reducing the drag of a 2-D bluff body is either to avoid the regular vortex shedding or to render it highly three-dimensional, thus reducing the vorticity concentration and the consequent kinetic energy level in the wake. This may be obtained by using splitter plates or modifications of the separation line that introduce a sufficiently high level of three-dimensionality.

Conversely, the present understanding on the flow and geometrical features influencing the drag of elongated axisymmetric bodies is certainly at a lower level. What is apparent is that in this case the pressure acting on the base (i.e. the body surface lying inside the wake) is approximately constant, at the same value that can be measured at the separation point; the latter is directly connected with the value of the velocity outside the boundary layer at the separation point. Therefore, one way of reducing base drag is to decrease the velocity outside the boundary layer at the separation point; this may be obtained, for instance, by appropriately modifying the geometry of the part of the body surface beyond its maximum cross-section, which is usually called “afterbody”.

In the following, a few methods for bluff-body drag reduction will be described in more detail, focusing the attention on those that may have a more direct bearing on the application to the aerodynamic design of heavy road-vehicles. In particular, methods for the increase of the base pressure will be considered, as base pressure drag is the main component of the drag of a bluff body. It should be noted that all the methods considered in the present section assume that the boundary layer be attached up to a rear separation point occurring at the contour of the body base. This obviously implies that the shape of the front part of the body be such that no global boundary layer separations occur; only local ones may be acceptable, provided reattachment occurs before the base.
2.2 Boat-tailing

Considering a 3D elongated bluff body with attached boundary layer up to a blunt, sharp-edged base perpendicular to the flow, its drag is directly connected with the value of the pressures acting on the base. This value is almost constant and equal to the pressure outside the boundary layer at the separation line (i.e. the sharp boundary of the base). Actually, the pressure starts decreasing before the base, which is then normally subjected to negative pressure coefficients (see Fig. 1a).

This trend of the pressure may be explained by considering the shape of the mean boundary of the separated wake (see Fig. 1b). Indeed, the mean streamlines bounding the wake tend to bend towards the axis after separation, and have thus a convex curvature in the separation region. This curvature is the responsible for the velocity increase that produces the decrease of the pressure in the zone before the base boundary.

A change of this flow and pressure configuration may be obtained through a geometrical modification denoted as boat-tailing, which consists in a gradual reduction of the body cross-section before the base. In Fig. 2 an example is shown of both the geometry of a boat-tail and of the drag reduction that may be obtained by increasing its length.
The mechanism that causes this effect is perhaps better understood by analyzing the typical pressure distributions along the boat-tail surface reported in Fig. 3 for different values of the Reynolds number (whose influence is rather small).

As can be seen, a significant reduction of the pressure occurs, which starts slightly before, and has a maximum slightly after, the beginning of the boat-tail. This corresponds to an acceleration of the flow just outside the boundary layer due to the convex curvature of the streamlines. Considering that the outward normal along the boat-tail surface has a positive component in the free-stream direction, this pressure decrease would produce an increase of the drag acting on the body. However, after its separation at the end of the body, the boundary layer bends in the opposite direction to form the boundary of the wake, and this causes a change of the curvature of the streamlines just outside it. Actually, the change to a concave curvature of these streamlines occurs before the end of the boat-tail, due to the increase of the boundary layer thickness, and this causes a rapid and significant decrease of the velocity and increase of the pressure. The effect of this pressure increase more than counterbalances the negative effect on drag of the previous acceleration, essentially because the pressure acting on the base (which, as already pointed out, is practically equal to the pressure at separation) is significantly increased with respect to the value that would be present without a boat-tail. Indeed, even positive pressure coefficients may be obtained, as is the case in Fig. 3. Obviously, for a good performance of a boat-tail in reducing drag, its length and the ratio between the final and initial diameter must be such that the boundary layer remains attached up to the base.
The commercial code FLUENT® was used by Del Duca (2005) to evaluate the aerodynamics forces acting on an axisymmetric body with different boat-tails, Fig. 4a). In spite of the fact that the base pressure distribution is not correctly predicted by a steady-flow RANS solver, it can be estimated from the pressure at the separation point on the lateral surface, which are well reproduced, as can be seen in Fig. 4b). Consequently, the drag variations due to boat-tailing could be evaluated with a more than satisfactory accuracy, as can be appreciated from Fig. 4c), where the numerical results are compared with the experimental data provided by E.S.D.U. (1996).
The idea of boat-tailing has been applied to reduce the drag of trucks, using various devices (see, e.g., Clark 2006, McCallen 2006, Ross 2006, and Fig. 5, in which a commercial inflatable device aimed at producing a boat-tailing effect is shown). However, in this application a major problem is avoiding a significant increase in the length of the trailer, which in general is not acceptable for both practical reasons and traffic regulations. Therefore, flaps of different shape are used (Fig. 6), which, even if shorter than would be necessary to produce a high drag reduction, often exploit another drag-reducing effect, viz. the presence of a more or less pronounced cavity in the base, thus providing significant reductions of the base drag (of the order of 5% to 10%).
It should be pointed out that, due to the above mentioned problems connected with the use of long boat-tailing devices, it might be interesting to study the possibility of considerably reducing their length (or increasing their drag-reducing performance for the same length) by coupling in their design moving surfaces (see section 2.4), which may allow the boundary layer to remain attached with boat-tail angles that would cause separation in the passive case.

2.3 Base Bleed

It has long been known that the blowing of flow through the base of a bluff body may significantly reduce its pressure drag, both for 2-D and 3-D bodies (see, e.g. Bearman 1966, 1967, Wood 1967, Leal & Acrivos 1969, Lewis & Chapkis 1969, Tanner 1975, Gai & Kapoor 1981, Delaunay & Kaisis 1999, Grosce & Meier 2001). The main mechanism causing this reduction is the alteration of the amount and distribution of the vorticity being introduced into the wake. In particular, if the blowing is through a hole in the center of the base, the consequence is the introduction of vorticity of opposite sign than that introduced into the wake from the boundary layers separating from the body surface and bounding the wake. But also if the blowing is through slots around the base contour or in various positions of the base surface, and if the outflow velocity is in an appropriate range, significant drag reductions may be obtained due to the increase in base pressure.

The fundamental parameters defining the configuration are the ratio between the hole or slot area and the base area, and the base bleed coefficient $C_q$, which is the ratio between the total flow through the hole or slots and the flow that would pass through the base area at the free-stream velocity.

An example of the effect of base bleed for a 2-D body is given in Fig. 7. As can be seen, the base pressure coefficient increases significantly with increasing blowing, but a maximum increase is reached for a certain value of the base bleed coefficient which depends on the diameter of the blowing hole. The reference steady flow value shown in the figure corresponds to the absence of vortex shedding in the wake, which may be obtained by using a splitter plate placed behind the body base.

Figure 7. Base pressure coefficient as a function of bleed coefficient (from Bearman 1966).
Curve 1: $d/h = 0.93$; curve 2: $d/h = 0.59$; curve 3: steady base flow.
It should be pointed out that the significant results obtained through base bleed in the 2-D case are strictly connected with the interference that it introduces with the alternate vortex shedding in the wake. Indeed, Bearman (1967) suggested that at high values of the bleed coefficient a recirculation zone forms immediately downstream of the body, and the regular vortex shedding is inhibited, with fluctuations possibly appearing more downstream. Flow visualizations were consistent with this interpretation. Thus, as already pointed out in 2.1, significant reductions of drag may be expected in the 2-D case due to the inhibition of the formation of the two rows of highly concentrated vortices in the wake, which introduce a high level of perturbation energy in the flow.

Nonetheless, even for axisymmetric bodies, in which the wake vorticity structures are generally less organized and concentrated, base bleed may give rise to drag reductions, probably because the kinetic energy of the wake is globally reduced by the presence of vorticity of different sign and by the reduced concentration of the vortical structures. A point to be noted is that the drag-reducing effect increases not only by reducing the blowing velocity, but also if the bleeding is through a porous surface, as can be seen in Fig. 8.

Figure 8. Effect of base bleed for an axisymmetric body with and without porous bleeding surface (from Sykes 1970). The curves are for different ratios between blowing area and base area.

The positive effects of base bleed have more recently been studied also numerically, either alone or in conjunction with boat-tailing. In any case, it should be recalled that blowing implies, in general, an additional power expense, which must be checked to be lower than the power saving due to the drag decrease. This is true even if passive ventilation through conduits connecting an air intake to the base is used, because of the additional pressure and friction drag inside the ducts. However, in this case a positive balance may be obtained (see, e.g., Suryanarayana et al, 1993, Suryanarayana and Prabhu 2000, Falchi et al. 2006), provided an adequate design is devised to reduce the inner flow drag while keeping the bleed parameters at favourable levels. This may be non-trivial, considering that in most practical applications a central hole from the front stagnation point to the centre of the base would not be allowed due to its interference with the use of the body volume, so that the ducts should connect an air intake in the
fore zone of high pressure to slots in the base, positioned either on its contour or in limited regions over its surface.

The application of base bleed through passive ventilation from limited regions of the base has recently been shown to be advantageous in the design of a high performance car (Lombardi et al., 2006). In that case the bleeding flow was obtained by diverting towards the base, at high car velocities, part of the air flowing in the cooling ducts. The effects of base blowing from slots of different dimension and placed in different regions of the car base were first studied numerically in order to obtain a preliminary optimization of the various parameters, and the subsequent road tests confirmed the improved performance of the finally chosen configuration.

It should be emphasized that, as was ascertained by Lombardi et al. 2006, wind tunnel investigations of the performance of base bleed using passive ventilation on scaled models may lead to incorrect conclusions. This derives from the significantly higher boundary layer thickness in the ducts at the lower Reynolds numbers of the tests with respect to the full-scale situation, which may lead to a much lower flow rate and, in general, to an underestimation of the drag-reducing effect.

2.4 Moving surfaces

The main objective of a control procedure is to prevent, or at least delay, the separation of the boundary layer from the wall. A moving surface attempts to accomplish this in two ways: it prevents the initial growth of the boundary layer by minimizing the relative motion between the surface and the free stream, and it injects momentum into the existing boundary layer. This has the consequence that the wall friction is reduced and the boundary layer is capable of remaining attached to the body surface for longer.

A convenient method of introducing a moving surface control is by means of rotating cylinders. Indeed, rotating cylinders need a negligible amount of power for their implementation, and thus they can be considered as a very promising device for practical applications. The effects of using rotating cylinders for controlling boundary layer is well known in literature, since Magnus in 1853 replaced the sail of a ship with a rotating cylinder. Goldstein (1938) investigated on the boundary layer control generated on a flat plate by setting a rotating cylinder at the leading edge of the body. In the ‘60s the use of rotating cylinders for increasing the high-lift characteristics of STOL aircrafts was studied in detail, and flight-tested in an existing aircraft. But the main amount of work on boundary layer control through rotating cylinders was carried out in the ‘90s by the research group of Modi (see, e.g., Modi 1997, Munshi et al. 1997, 1999, Modi and Deshpande 2001).

At a first stage they investigated the effects of rotating cylinders on the maximum lifting performance of airfoils, and Fig. 9 shows some of their results, for different positions of the cylinders and different ratios between the cylinder surface velocity and the free-stream velocity.

The physical mechanism producing the increased lifting performance of the airfoils may be appreciated from the flow visualizations of Fig. 10, which also clearly demonstrate the importance of the value of the cylinder surface velocity.
Figure 9. Lift curves for airfoils with different moving cylinders devices (from Modi 1997).

Figure 10. Flow visualizations for airfoils with different cylinder velocities (from Modi 1997).
Figure 11. Flow visualizations and comparison with numerical results for a flat plate with cylinders rotating at various velocities (from Modi 1997).

Figure 12. Drag curves for a flat plate with rotating cylinders (from Modi 1997).
The attention of Modi and his group was then turned to bluff bodies, for which a reduction of the pressure drag is more important than the increase in lift. In this case the positioning of rotating cylinders at the edges of the body delays separation and progressively reduces the width of the wake with increasing cylinder velocity. The effects on the flow field for a flat plate are shown in Fig. 11, while the relevant drag curves are reported in Fig. 12. The drag coefficients obtained by positioning two cylinders at two of the edges of a rectangular 2-D prism are shown for different directions of the free-stream and cylinder rotation velocity in Fig. 13. As can be seen, in all cases definitely remarkable drag reductions may be obtained, provided the surface cylinder velocities are sufficiently high and the flow is attached before the downstream cylinder.

Figure 13. Drag curves for a 2-D prism with rotating cylinders at its edges (from Modi 1997).
Figure 14. Sketch of a tractor-trailer model with rotating cylinders (from Modi 1997).

Figure 15. Drag results for a tractor-trailer model with rotating cylinders (from Modi 1997).

Variation of the drag coefficient $C_D$ with the speed ratio for spline twin-cylinder configuration:
(a) Case 1: both cylinders flush; (b) Case 2: front cylinder flush, rear cylinder raised 6-35 mm. Re = $10^5$.

(*$U_{ef}/U = 0$; (G) $U_{ef}/U = 1.4$; (x) $U_{ef}/U = 2.7$; (Q) $U_{ef}/U = 4.1$; (h) $U_{ef}/U = 6.1$.)
Finally, wind-tunnel tests were performed to verify the effectiveness of rotating cylinders in reducing the drag of trucks. A tractor-trailer scale model fitted with two rotating cylinders over the trailer was used (Fig. 14), and it was found that the surface roughness of the cylinders was also important to improve their performance, so that machining helical or spline grooves on the cylinders could be favourable. A substantial drag reduction was observed by varying the rotational speed of the cylinders as reported in Fig. 15. These results are significant, considering the limited range of action of the cylinders and the shape of the model, which presumably induces large flow separations.

More recently, the good performance of rotating cylinders in altering the flow and reducing bluff body drag was also demonstrated by the experimental and numerical investigations of Singh et al. (2005) and Beaudoin et al. (2006).

In conclusion, the use of moving surfaces, and particularly rotating circular cylinders, seems to be an extremely promising means of controlling boundary layer separation, and thus reducing the drag of heavy vehicles. Incidentally, the power requirement that is needed for the rotation of the cylinders is probably limited, and almost certainly lower than what is necessary for active suction or blowing. Furthermore, moving surfaces might be used in conjunction with other drag-reducing techniques, as for instance boat-tailing. Nonetheless, apart from possible practical difficulties of application, the effectiveness of this methodology depends on a high number of parameters. Therefore, dedicated investigations may be necessary, with a previous clear identification of the applications to be considered and of the design goals.

2.5 Manipulation of boundary layer conditions

There are several indications in the literature (Rowe et al. 2001, Whitmore & Naughton 2002, Durgesh & Naughton 2004) that the increase in boundary layer thickness may decrease the drag of two-dimensional bluff bodies with a blunt base, by increasing the base pressure; an example of this effect is shown in Fig. 16.

![Figure 16](image.png)

Figure 16. Variation of base pressure and Strouhal number with boundary layer displacement thickness for an elongated two-dimensional bluff body (from Rowe et al. 2001).
Also the vortex shedding phenomenon (which is typical of 2-D bodies) is affected by the variation of the thickness of the boundary layer, as is witnessed by the variation in Strouhal number (i.e. the non-dimensional frequency of vortex shedding) reported in Fig. 16; however, it is not yet clear if the increase in base pressure is or not strictly connected with this influence on the vortex shedding characteristics.

In any case, a similar effect was also found by Tanner (1985) considering the base pressure behind a backward facing step in supersonic flow (in which vortex shedding is not present), by Porteiro et al. (1983) in a subsonic flow behind an axisymmetric blunt base, and by Buresti et al. (1997), who studied the base pressure of an axisymmetric bluff body with different boundary layer conditions.

In particular, Porteiro et al. (1983) suggest that the significant parameter influencing the base pressure is the momentum thickness of the boundary layer at separation. Their experiments were actually not carried out using a bluff body, but rather a blunt-based sting placed at the exit of a nozzle (see Fig. 17a), fitted with a porous base plate, through which blowing and suction could also be introduced; thus the effects of positive and negative base bleed could also be studied. Their results (see Fig. 17b, in which the base bleed coefficient is designated as $\Gamma$) suggest that the effect of increasing the boundary layer thickness may be even higher than that of base bleed.

Figure 17. Variation of base pressure with boundary layer momentum thickness and base bleed ($\Gamma$=$C_q$).

(From Porteiro et al. 1983)
Even if the amount of dedicated investigations in the literature is still limited, so that the details of the phenomenon are not yet completely understood, the importance of the momentum thickness seems to be confirmed also by the measurements carried out by Britcher and Alcorn (1994) on a slanted-base body. Therefore, it would be interesting to study in more detail the effects of changing the thickness and characteristics of the boundary layer before separation on a bluff body with a blunt base and attached boundary layer over its lateral surface.
3 Further drag-reduction methods typical of tractor-trailer configurations

3.1 Tractor-trailer gap

The gap between the tractor and the trailer is surely of utmost relevance for the pressure drag of this type of heavy vehicles. This gap is obviously necessary for the manoeuvrability of the trailer, and its length is usually minimized to reduce the entrainment of the flow into the gap, which would impinge on the frontal area of the trailer, increasing the drag. Typical devices used for the reduction of the pressure drag associated with this region of the truck are cab extenders, shown in Fig. 18a), which may be used for both the vertical and upper edges of the tractor base. The effectiveness of these devices for reducing the drag is largely documented in the literature (see, e.g., Ross 2006), and their main objective is reducing the gap flow. This goal can also be achieved by closing the gap through a suitable rotational flow by using a splitter plate, as shown in Fig. 18b). Another type of device for the control of the gap flow is the “vortex trapping” system proposed by Wood (2004), and shown in Fig. 19.

![Figure 18. Devices for reducing the pressure drag due to the tractor-trailer gap: a) cab extenders; b) splitter plate (from Salari 2006).](image)

![Figure 19. Trapping vortex device (from Wood 2004).](image)

It should be pointed out that all these devices have generally also a favourable effect in non-symmetrical flow conditions, i.e. when a cross-wind is present; furthermore, they might be coupled with adequate blowing from both the tractor and trailer boundaries to increase the drag-reducing effect.
3.2 Fairings and flow-deflection devices

The geometry of the undercarriage of a truck is generally considerably complex, with various types of bluff bodies that cause flow separations, thus leading to highly unsteady flow conditions. The fairing of the bodies present on the undercarriage area (e.g. axles of the wheels) is thus a possible means to reduce the drag of a tractor-trailer configuration. For instance, a fairing may be positioned either in front (Fig. 20a), or behind (Fig. 20b) the rear axle of the trailer to reduce the pressure drag not only of the axle but also of the trailer base, by deviating the flow and reducing the size of the wake.

![Figure 20. Fairing of the rear axle of a trailer: a) front fairing; b) rear fairing (from Wood 2004).](image)

Other types of fairings were developed for the undercarriage, as devices for splitting the flow in front of the rear axles, or lateral skirts for controlling the cross-flow below the trailer (Fig. 21). All these devices may have an effect (favourable or non-favourable) on the spray of water over neighbouring vehicles in raining conditions.

![Figure 21. Various types of fairings for the undercarriage flow (from Salari et al. 2006).](image)

It should be noted that by combining undercarriage fairings with an accurate positioning of moving surfaces for boundary layer control a stronger drag reduction might possibly be achieved.
3.3 Wheel-house shaping

The wheels of road vehicles are, in general, a source of considerable aerodynamic drag. However, a judicious choice of the geometry of the wheel-houses may favourably change the aerodynamic loads, by delaying the flow separation and/or producing required components of the pressure forces. For instance, wheel-house shaping has been successfully applied in the design of high-performance cars (see, e.g., Buresti 2005) in order to reduce drag and, specially, to increase the downforce, which is essential in this type of vehicles.

However, the use of this technique to reduce the drag of heavy road-vehicles poses much greater problems, due the fact that the flow that impinges on the wheels generally originates from upstream separations, and is thus (at least in part) a highly unsteady wake flow. Nonetheless, devices may be devised that, apart from a possible drag reduction effect, may also produce a favourable outcome as regards the “splash and spray” phenomenon (see Fig. 22).

Figure 22. Example of faired wheel-houses.

In spite of the above-mentioned difficulties, which are a challenge for both numerical and experimental investigations, the development of possible methods for the control of the flow in front, around and behind the wheels remains one of the important points worth of further research, due to its significant influence on the local and global forces acting on a tractor-trailer configuration.
4 Overview of the possible activities

In the following, a brief description is given of the possible research activities for the reduction of the drag of heavy-duty tractor-trailer trucks that were proposed by DIA within the present contract. All the activities are based on the use of numerical codes.

Aerodynamic optimization of the tractor

The optimization of the tractor shape is important both for the reduction of its drag and for the control of the interference it generates with the flow around the trailer.

In general, the shape of the front part of the tractor should be such as to avoid or reduce to a minimum flow separations, and minimize its pressure drag. This normally implies a rounding of the various surfaces, as, e.g., the front hood, the windshield and the bumper. Furthermore, it is important that the basic shape of the rear boundary of the tractor be such that the flow separating there reattaches on the lateral surfaces of the trailer.

However, a particular attention should be devoted to certain details, like the A-pillars and the mirrors, which are a source of flow separation, high local drag and significant interference with the downstream portion of the vehicle.

Boat-tailing

As discussed in section 2.2, the effects of boat-tailing is well documented. Therefore, some dedicated research activity might be envisaged in order to optimize boat-tailing devices that may be applied to the contour of a truck trailer.

However, as already mentioned, instead of further analysing boat-tailing flaps, it is considered preferable to dedicate an activity in the present research program to study if favourable effects may be obtained by coupling these devices with moving surfaces.

Moving surfaces

As already pointed out in section 2.4, the use of moving surfaces, and in particular rotating cylinders, seems to be a promising means for significantly altering the flow conditions and reducing the drag of bluff bodies. Therefore, their use for the modification of the flow in various parts of a tractor-trailer configuration may be envisaged. However, a significant basic research activity is needed, as the influence of the various parameters is not yet completely established.

In particular, it would be interesting to study if pressure drag decreases can be achieved by positioning rotating cylinders at different locations of the truck as:
- Along the rear edges of the trailer, over its lateral surfaces or around the base surface;
- Along the front edges of the trailer;
- Along the rear edges of the tractor;
- In any position where a momentum injection in the boundary layer might be considered to be favourable.

However, another possible use of rotating cylinders is for enhancing the performance of other drag-reduction methods, and in particular boat-tailing flaps.

A detailed research program devoted to this particular activity will be described in section 5.1.
**Boundary layer manipulation**

The objectives of an activity for the manipulation of the boundary layer over the truck trailer may be twofold:
- reducing the friction drag over the trailer surface;
- reducing the base drag of the trailer.

Both these objectives pose non-trivial problems of reliability of the numerical simulations that use commercial RANS codes, even if these difficulties seem to be greater for the first one.

Indeed, as regards the friction drag, leaving out the study of the effects of surface films, like riblets or similar (for the practical impossibility of the numerical simulation of their effect), and of methods for boundary layer suction (due to their complexity and poor practical feasibility), the only possibility seems to be analysing the advantages that might derive from the introduction of surface elements producing local (and even extensive) boundary layer separations. However, for these devices to be effective, it is necessary not only that their added drag be lower than the friction drag reduction they provide, but also that the flow may surely reattach before the rear part of the trailer surface (in case, using further elements for the purpose). Anyway, apart from the limited reliability of RANS codes for the evaluation of friction drag, it should be pointed out that an acceptable assessment of the flow behaviour in separated regions would probably require using the much more expensive unsteady simulations.

The situation is slightly more favourable for the analysis of methods for the reduction of the base drag by changing the thickness and characteristics of the boundary layer, already discussed in section 2.5. Indeed, provided the numerical simulations are carried out using an adequately refined (and possibly structured) grid, it may be possible to study the effect of introducing different types of roughness elements over the rear part of the trailer surface. Even if the optimum roughness elements should probably be three-dimensional, it might be useful to start with simple cross-flow small bars, and then, if promising results are obtained, analyse the effect of more complex shapes.

**Base bleed**

The effects of base bleed may be studied numerically in a relatively simple way, even if the number of parameters to be taken into account may be significant, so that a high number of runs may be necessary to optimize a particular application.

As regards the drag reduction of tractor-trailer configurations, base bleed may be used for several purposes. The first, and most obvious, is the reduction of the base drag of the trailer; in this case a research activity might be devoted to studying the possibility of having favourable effects by blowing air through only limited regions, like the boundary of the base, which would minimize the added complexity introduced in the design.

Another possibility is using base bleed for the control of the gap flow between the tractor and the trailer.

Finally, base bleed may be used for the drag reduction of the mirrors. In this case, it might also be possible to advantageous use passive ventilation in order to avoid the necessity of introducing of an “ad hoc” blowing system.
**Aerodynamics of the tractor-trailer gap**

The effects of different devices to influence the flow in the tractor-trailer gap may be studied numerically with the objectives of reducing both the drag and the cross-wind sensitivity of the configuration.

Indeed, most of the information present in the literature on the performance of these devices (see section 3.1) typically refer to U.S.A tractor-trailer configurations, which are generally characterized by wider gaps than is the case for the European ones; therefore, the extrapolation of the results to the latter vehicles is not immediate, and specific investigations are needed.

The most promising devices that may be analysed (after a characterization of the flow for the basic configuration) are tractor-cab extenders, splitter plates and various types of rounding of the trailer front edges. Furthermore, an interesting method whose performance might be investigated is the (active or passive) blowing of air from various positions around the tractor base or the trailer face surface.

**Fairings of the trailer undercarriage**

In spite of the difficulties that the complex undercarriage flow poses for a numerical simulation, the possible drag-reduction potential of a better control of this flow suggests that investigations in this field might be advantageous. Several different devices may then be studied, both for shielding high-drag components (like the wheel axles), and for controlling the flow entering into or exiting from the undercarriage region. On this respect, particularly important is the flow from the undercarriage to the base, as it may significantly alter the base pressure, and thus the global drag.

Once again, fairings and shielding flaps may be advantageously coupled to other flow-control methods, like blowing or the insertion of moving surfaces, in order to enhance their performance.

However, the investigations in this field would probably require significant computational resources and the possibility of a strict synergy with wind-tunnel investigations.

**Aerodynamics of wheels**

The importance of improving the present understanding and control of the flow around the wheels of vehicles in general, and trucks in particular, cannot be overestimated. Indeed, not only are the wheels high-drag items, but they also have a significant influence on the undercarriage and base flows.

Therefore, the study of wheel fairings and/or carefully designed wheelhouses is a possible important research activity. However, as is the case for the analysis of the undercarriage aerodynamics, due to the complexity of the flow conditions significant computational and temporal resources may be necessary to achieve satisfactory results.
5 Description of the chosen activities

The research activities that were jointly agreed by CRF and DIA as the object of the present contract will now be described in more detail.

5.1 Analysis of the aerodynamic features of a typical tractor-trailer

A complete analysis of the flow over a typical tractor-trailer configuration, and of the consequent aerodynamic loads, will be carried out. The mathematics of the considered geometry will be provided by CRF, and DIA will carry out the numerical work that is necessary for its implementation as an input for the fluid-dynamic code.

The main objectives of this activity are to obtain an estimate of the contribution of each element of the body to the total drag, and to check the general quality of the flow field, and, in particular, the presence and extent of separated flow regions over the surface of the trailer. This will permit to identify the most significant items to be considered for the reduction of the aerodynamic drag, and thus to obtain important clues for the present and future research activities.

5.2 Development of devices comprising rotating cylinders

This activity will be focused on the evaluation of the possible effectiveness of rotating cylinders as a means for reducing the pressure drag of a tractor-trailer configuration.

In order to achieve an adequate understanding of the influence of the various parameters that influence the performance of different devices using circular cylinders, it is necessary to carry out a detailed basic investigation requiring a significant number of computational runs. Therefore, the analysis will be performed using a simplified body, which, even if meaningful from the point of view of a subsequent possible application to a tractor-trailer configuration, will allow rapid simulations to be carried out in order to investigate on the effects of the variation of the numerous geometrical parameters.

The main part of the body will be a prism with a rectangular cross-section having the side ratio of a typical trailer, and which will be smoothly connected to an ellipsoidal forebody with the same ratio of the semiaxes. Various preliminary runs will be carried out in order to optimize the grid (which will be structured at least in the region near the surface), and to characterize the main flow features. In particular, it must be ascertained that the boundary layer be attached over the lateral surfaces of the body, before separation at the sharp base contour.

Subsequently, runs will be carried out to study the aerodynamic behaviour and the load variations of the two basic configurations shown in Fig. 23. In both cases the main parameters whose influence must be investigated are the cylinder diameter, position and rotation velocity. As regards the cylinder dimensions, their full-scale diameters will in any case be kept below 100-200 mm in order to assure the feasibility of an application to a real truck.

It should be pointed out that there are no available data in the literature on this particular type of configurations, and that a positive drag-reducing effect (particularly as regards case 2 of Fig. 23) may be expected only if the decrease in base drag is larger than the predictable added contribution of the cylinder surface to the pressure drag.
The performance of innovative configurations, in which rotating cylinders are combined with a boat-tailing flap, will then be analysed. The main objective of this activity is to analyse the possibility of reducing the length of boat-tailing devices (or increasing their drag-reducing performance for the same length) by coupling rotating cylinders in their design; the idea is that with the cylinders the boundary layer might remain attached even with boat-tail angles that would otherwise cause its separation.

The two basic configurations that will be considered at this stage are shown (only schematically) in Fig. 24.

Further parameters whose variation should be studied for these configurations (besides the diameter, position and velocity of the included cylinders) are the shape of the fairing, and the value of its rear boat-tail angle $\beta$. For configuration 3 the maximum allowed length will be 400 mm.

Possible combinations of the shapes of Fig. 24, like those shown in Fig. 25, will be analysed only if possible within the time limits of the present research program, and if the results for the basic configurations are considered to be sufficiently positive.

In view of the necessity of deeply investigating the actual drag-reducing potential and practical applicability of these devices, a high number of different configurations will probably have to be analysed. Therefore, this activity is scheduled for the whole duration of the research project.
5.3 Manipulation of trailer boundary layer for base-drag reduction

Due to limited information present in the literature on how the variation of the characteristics of the boundary layer at separation may influence the base pressure drag of a bluff body (see section 2.5), also this activity will have a basic research character. Therefore, the simplified bluff body described in the previous section will be used also for studying this subject.

The analysis will be carried out by introducing boundary layer manipulation devices, starting from simple small bars, perpendicular to the flow direction, positioned over the lateral surfaces of the body. In any case, to limit their added drag, the height of these devices must be of the order of the boundary layer thickness, which, obviously, must be adequately resolved in the numerical simulations.

The parameters that will have to be studied are then the number, position and size of the bars, and their effect on the various thickness parameters of the boundary layer and on the pressure acting on the body base.

If the results of the first tests are sufficiently positive, unsteady RANS simulations may be carried out for the best obtained conditions, in order to check if a more accurate description of the wake flow field may have an effect on the base pressure distribution. Furthermore, more complex three-dimensional shapes may also be considered for the manipulation devices.

5.4 Aerodynamic optimization of the mirrors

The mirrors give a non-negligible contribution to the overall drag of heavy vehicles due to their considerable cross-flow dimension and to the high values of their drag coefficients. In the present research program an activity will then be devoted to the reduction of the drag of a typical heavy-duty truck mirror.

Requirements and specifications for the mirrors will first be provided by CFR; in particular, minimum and maximum size of the mirrors, their orientation with respect to the free-stream direction, dimensions of the holders, constrains on the mirrors position.

An aerodynamic optimization of the basic shape of the mirrors will then be performed. In this analysis, in order to mimic the actual flow field in which the mirror is placed, a body approximately reproducing the main geometrical features of the tractor cab will be considered in the simulations (see Fig. 26).
Figure 26. Sketch of the simplified cab shape for the computational analysis of the mirrors.

The optimization will be devoted to devising a shape of the fore part of a bluff body (which corresponds to the front fairing of the mirror) which, for given dimensions of the body base, minimizes its total drag (and/or the size of its wake). Obviously, the constraints on the maximum global dimensions of the mirror must be taken into account in the procedure.

Subsequently, an investigation will be carried out on the potential of using active base bleed for obtaining a further reduction of the mirror drag. Clearly, the air blowing may be introduced only through slots placed around the mirror surface (or surfaces, if more than one is present), and this may imply a certain increase of the considered bluff body base with respect to the size of the mirror surfaces. The parameters to be analysed in this phase are the position and dimensions of the slots, and the value and direction of the bleeding air velocity. The problem of the supply of the bleeding air will not be taken into consideration at this stage.

Finally, and particularly if the results of the previous activity are positive, a passive bleeding configuration will be analysed. To this end, the optimum shape of the slots and direction of the bleeding air obtained in the previous stage will be used, and a body will be designed with an air intake in its front part and adequate inner ducts leading to the base slots with the appropriate direction.

This last stage is rather complex, due to the large amount of geometrical parameters that may influence the performance of the passive ventilation (viz., all those defining the dimensions and shape of the air intake and of the inner ducts). In practice, it is necessary to define the geometry of two bodies, the outer shield (with the air intake) and the inner shield, which together form the boundaries of the duct conveying the air from the intake to the slots in the base.

Therefore, it is possible that only a preliminary assessment of the technique may be carried out in the limited time that is presently foreseen for this activity (see 6). Nevertheless, in case very promising results are obtained, CRF and DIA may jointly agree to adequately modify the present program in order to allow a more detailed optimization of the mirror configurations to be carried out.
5.5 Control of tractor-trailer gap flow through blowing.

As already pointed out, the type of flow existing in the tractor-trailer gap is particularly important, not only because that particular region gives a significant contribution to the global drag of the vehicle, but also because it influences the flow over the trailer, and thus its friction and pressure drag. Furthermore, the sensitivity of the truck to cross-wind may be considerably affected by different configurations of the tractor-trailer gap region, due to larger or lower quantities of free-stream flow entering this region and thus impinging over the front surface of the trailer.

In the present research program an activity is foreseen to study if the flow in the tractor-trailer gap region may be favourably controlled by means of blowing through appropriate slots, positioned either over the rear surface of the tractor or over the front surface of the trailer.

The first stage of the activity is the critical analysis of the gap flow present in the basic configuration used for the simulation described in 5.1, in order to identify the regions where the blowing of air may be more effective. Subsequently, different configurations will be analysed, in order to analyse the effect of the variation of the shape, position and dimensions of the slots, and of the direction and intensity of the bleeding flow.
## 6 Temporal program of the activities

<table>
<thead>
<tr>
<th>Activity</th>
<th>Sept</th>
<th>Oct</th>
<th>Nov</th>
<th>Dec</th>
<th>Jan</th>
<th>Feb</th>
<th>Mar</th>
<th>Apr</th>
<th>May</th>
</tr>
</thead>
<tbody>
<tr>
<td>Literature survey</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Global aerodynamic analysis</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rotating cylinders</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Boundary layer manipulation</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mirrors</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tractor-trailer gap blowing</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Final Report</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Interim Reports</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
References