

Structures of Flow Separation on a Passenger Car

Sabine Bonitz, Lars Larsson, and Lennart Lofdahl
Chalmers University of Technology

Alexander Broniewicz
Volvo Car Corporation

ABSTRACT

The phenomenon of three-dimensional flow separation is and has been in the focus of many researchers. An improved understanding of the physics and the driving forces is desired to be able to improve numerical simulations and to minimize aerodynamic drag over bluff bodies.

To investigate the sources of separation one wants to understand what happens at the surface when the flow starts to detach and the upwelling of the streamlines becomes strong. This observation of a flow leaving the surface could be captured by investigating the limiting streamlines and surface parameters as pressure, vorticity or the shear stress.

In this paper, numerical methods are used to investigate the surface pressure and flow patterns on a sedan passenger vehicle. Observed limiting streamlines are compared to the pressure distribution and their correlation is shown. For this investigation the region behind the antenna and behind the wheel arch, are pointed out and studied in detail.

Besides the discussion of the correlation between limiting streamlines and the surface pressure distribution, it is discussed how the surface pressure and limiting streamline development is formed. It is shown how vortices emanating from the antenna influence the surface pressure and therefore the limiting streamline pattern. Behind the front wheel arch it is explained how the separation bubble upstream influences the development of the limiting streamlines further downstream.

CITATION: Bonitz, S., Larsson, L., Lofdahl, L., and Broniewicz, A., "Structures of Flow Separation on a Passenger Car," *SAE Int. J. Passeng. Cars - Mech. Syst.* 8(1):2015, doi:10.4271/2015-01-1529.

INTRODUCTION

Reduced fuel consumption is one of the important goals in the aerodynamic development of passenger vehicles. For bluff bodies the overall drag is dominated by pressure drag. In order to decrease aerodynamic drag it is desired to reduce pressure drag, by reduction and minimization of separation regions. Especially for complex bluff bodies like road vehicles it is important to improve the understanding regarding driving forces and origins of separation.

Contrary to investigations over a flat plate or basic geometries, the surface of vehicles is usually curved and the flow over the body is influenced by different flow conditions, as well as the interaction and interference of different flows. That makes it difficult to identify and isolate a particular type of separation, as an interaction of several types of separation is observed.

Missing criteria for three dimensional separation points complicate the prediction of separating flow. However a characterization and the description of the flow field is possible.

The topological theory describes different types of separation by means of singular points. These can be visualized by showing the limiting streamlines. An investigation of the corresponding surface parameters (e.g. surface pressure) can give a correlation between the separation and physical parameters.

The velocity field as a continuous vector field can be described by means of critical points as it is described for instance in [1, 2, 3, 4, 5, 6, 7]. In vehicle aerodynamics this method was used to characterize occurring flow structures and to derive the flow development [8, 9, 10, 11]. Also the influence of the backlight angle, deck lid length or C Pillar length and radius onto the flow pattern was shown by many researchers, for instance [12, 13, 14, 15, 16]. But how these patterns are formed and under which conditions they occur is still an insufficiently answered question.

This underlines the importance of an understanding of the physics behind three-dimensional flow separation in order to be able to decrease aerodynamic drag further.

In [17] the flow pattern with its limiting streamlines is described for the upper body of a sedan passenger car. It is shown that the flow around a shark fin antenna obeys topological rules and it describes how the pressure gradient influences the development of limiting streamlines.

The general objective of the present project is to understand the details of the flow around the upper body of a passenger car. Ultimately, such knowledge will lead to a better aerodynamic design with less drag. The present paper presents the first part of the investigation where emphasis is placed on two particular regions. First the flow behind the antenna will be presented and the occurring vortices are connected to the pressure distribution on the surface. Second the flow behind the front wheel arch will be described and again, the limiting streamline development and their connection to the surface pressure distribution will be explained.

BACKGROUND

Chapman and Yates introduce in their work [1] three different types of separation.

- 1). Type I separation - "bubble separation"
- 2). Type II separation - "horn separation"
- 3). Crossflow separation

The types of separation and the application of certain topological rules were discussed in detail in [17]. The following figures (Figure 1, Figure 2, Figure 3) show the topology of the different separation pattern on a body of revolution. Nodes of attachment (N_a) are singular points from which streamlines emanate. Nodes of separation (N_s) are singular points where streamlines meet and leave the surface. S_s describes saddle points of separation and S_a saddles of attachment.

Looking into cross planes, only two velocity components can be displayed. Therefore so-called half saddles (S') and half nodes (N') are introduced.

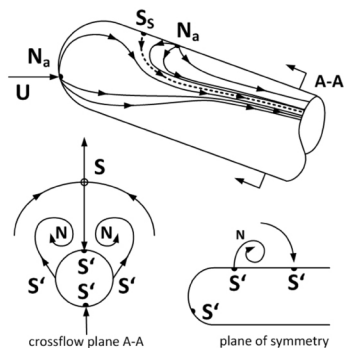


Figure 1. Type I separation [17]

Type I separation originates at a saddle point of separation and a separation line (--) divides the body surface into two regions. A limiting streamline originating at the node of attachment (N_a) to the left cannot enter the region on the other side of the streamline originating at S_s . This streamline is a separation line where the fluid

rolls up into a vortex with the center N (crossflow plane A-A). The node of attachment N_a to the right is necessary to allow the fluid to enter the separated region.

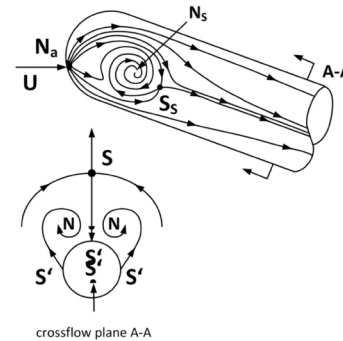


Figure 2. Type II separation [17]

The second type is also called horn separation. It again originates from a singular point SS , but the body is not divided and streamlines from N_a can enter all regions on the surface. For this type a vortex leaves the surface at N_s .

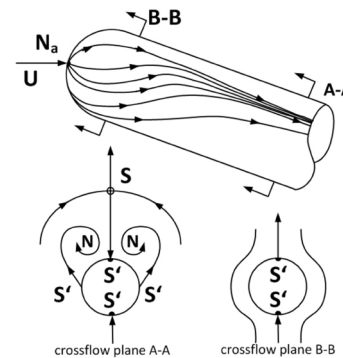


Figure 3. Crossflow separation [17]

The crossflow separation does not start from a singular point. In fact, the starting point cannot be localized exactly. The separation is characterized by converging streamlines, which leave the surface and roll up into a vortex.

METHOD

Test Object and Numerical Model

The test object was a Volvo S60 sedan passenger car. For the numerical simulation a full scale model with a detailed underbody was used as can be seen in Figure 4

The CFD simulations were done in Star CCM+ v8.04 and post processed in ENSIGHT 10.0.3. For the investigation a steady state RANS calculation with a realizable k-ε turbulence model was used, based on the work of [18].

The mesh size on the surface had a maximum length of 7mm with refinements down to 2mm. The volume mesh had 3 prism layers with a total height of 7mm. The resulting y^+ values were in the range of 30 and 150. The investigated case had 96 million volumetric cells.

The wheel rotation was simulated with a moving wall boundary condition and MRF (multiple reference frame) method between the spokes.

Stable residuals below 10^{-3} as well as a drag and lift coefficient variation below 0.5 counts for the last 500 iterations were the conditions for a converged simulation.

The focus of this investigation lies in the analysis of the limiting streamlines and their correlation to the surface pressure. This analysis is based on numerical calculations.

It is known that this method does not describe the flow field in all its details. For instance, no transient information is considered and wall functions are used to model the near wall flow field. This solution with its assumptions is however used to get a better understanding of the connection between pressure distribution and limiting streamlines.

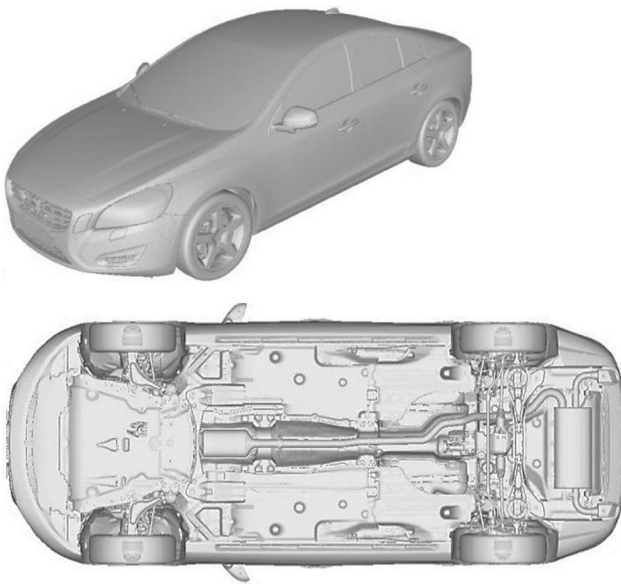


Figure 4. Numerical S60 model

Comparison to Experimental Results

The usage of steady state simulations is very common in industry. Therefore a RANS solution was chosen to be investigated, before looking at more advanced solutions.

Experimental investigations in [17] showed that the measured and calculated pressure distributions match very well. This can also be seen in Figure 5.

In an additional test the pressure distribution on the side of the car (behind the wheel arch) was measured (Figure 6b). 124 pressure holes were distributed over the surface. The pressure measurements were time averaged over 30 s.

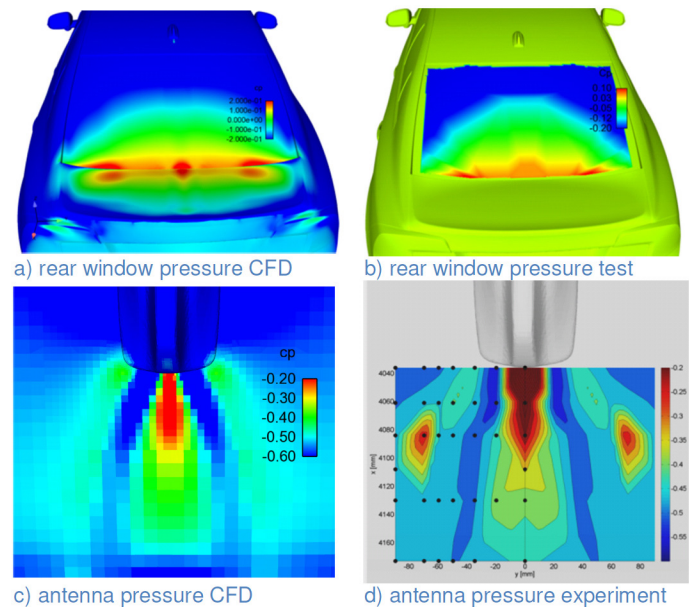


Figure 5. Calculated and measured pressure distribution for the rear window and antenna [17]

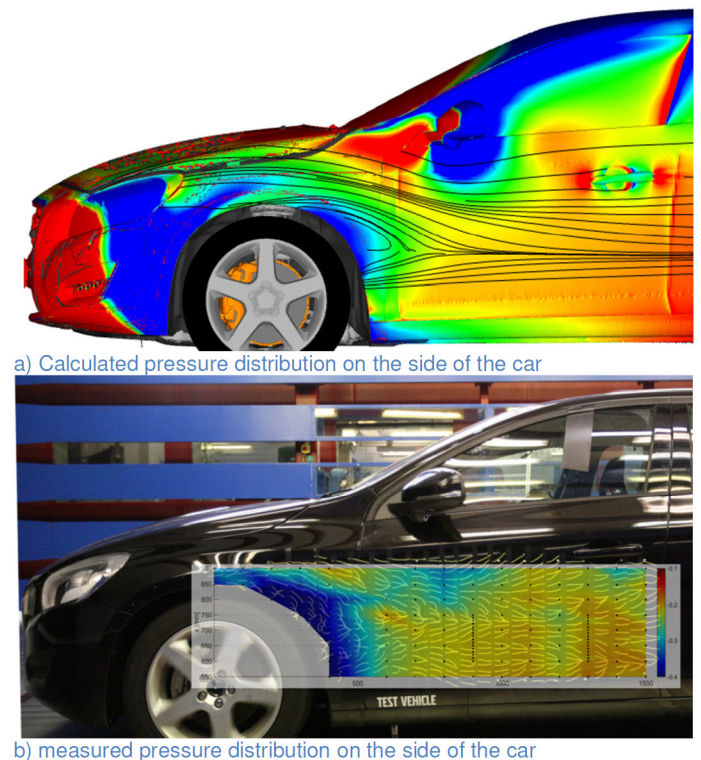


Figure 6. measured and calculated pressure distribution behind the wheel arch

Figure 6 shows numerical and experimental pressure distribution behind the wheel arch. The low pressure region around the wheel arch seems to be stronger in the wind tunnel test. In both cases the low pressure influence from the side mirror can be observed and also the overall pressure distribution is similar.