

Performance of an Automotive Under-Body Diffuser Applied to a Sedan and a Wagon Vehicle

Jesper Marklund
Combitech AB

Lennart Lofdahl
Chalmers Univ. of Technology

Hakan Danielsson
Chalmers Univ of Technology

Gunnar Olsson
LeanNova Engineering AB

ABSTRACT

Reducing resistance forces all over the vehicle is the most sustainable way to reduce fuel consumption. Aerodynamic drag is the dominating resistance force at highway speeds, and the power required to overcome this force increases by the power three of speed. The exterior body and especially the under-body and rear-end geometry of a passenger car are significant contributors to the overall aerodynamic drag. To reduce the aerodynamic drag it is of great importance to have a good pressure recovery at the rear. Since pressure drag is the dominating aerodynamic drag force for a passenger vehicle, the drag force will be a measure of the difference between the pressure in front and at the rear. There is high stagnation pressure at the front which requires a base pressure as high as possible. The pressure will recover from the sides by a taper angle, from the top by the rear wind screen, and from the bottom, by a diffuser. It is not necessarily the case that an optimized lower part of the rear end for a wagon-type car has the same performance as for a sedan or hatch-back car. This study focused on the function of an under-body diffuser applied to a sedan and wagon car. The diffuser geometry was chosen from a feasibility stand-point of a production vehicle such as a passenger car. The fluid dynamic function and theory of the automotive under-body diffuser working as a drag reduction device is discussed. The flow physics of the under-body and the wake was analyzed to understand the diffuser behaviour in its application to lift and drag forces on a vehicle in ground proximity. This work is mainly a numerical analysis that uses the traditional CFD approach from the automotive industry. Results from this study show a potential to reduce aerodynamic drag of the sedan car approximately 10%, and the wagon car by 2-3 %. The possible gain was much bigger for the sedan vehicle and the optimum occurs at a higher diffuser angle. This was most likely due to the fact that the sedan car in its original shape produced more lift force than the wagon, a wagon usually produces very little lift or even down-force. Lift forces were also reduced with the use of under-body covers with diffuser. The down-force increased, or lift force decreased, linearly with increased diffuser angle, and the trend was the same for both sedan and wagon rear ends. Flow analysis of the wake showed the importance of how the wake is balanced.

CITATION: Marklund, J., Lofdahl, L., Danielsson, H. and Olsson, G., "Performance of an Automotive Under-Body Diffuser Applied to a Sedan and a Wagon Vehicle," *SAE Int. J. Passeng. Cars - Mech. Syst.* 6(1):2013, doi: 10.4271/2013-01-0952.

INTRODUCTION

Reducing the fuel consumption is one of the main goals in the field of vehicle aerodynamics. Controlling Lift forces is

very important for vehicle stability and generally there is a desire to maintain as low lift forces as possible. Lift and drag have a very complex relationship for bluff bodies, but for simplified bodies there is a parabolic relationship with an

optimum drag relative to lift (1). Aerodynamic drag arises from energy losses in the flow behind a body that leads to a difference in pressure before and aft the body. There will be stagnation pressure at the front, and ideally the pressure would recover to the same level in the rear to give no drag. This will never happen due to friction and separation that give energy losses that leads to losses in pressure recovery. Lift forces normally generate longitudinal vortices due to pressure differences between the upper- and under-side that generate drag due to energy losses in the flow related to the vortices. Wing theory tells us that this vortex induced drag increases with the square of lift and a constant, dependent on the aspect ratio (length/width) of the wing. There is a similar relationship for bluff bodies (2) but not easily distinguished due to the vortices mixing and creating turbulence.

The largest part of the aerodynamic drag force of a bluff body is related to the rear end and the wake flow. The exterior body, underbody, wheel-house and cooling system are the major parts that generate drag. One way to divide how much the different areas contribute to the total drag force is suggested by BMW: that 20% comes from the under-body, 30% from the wheels and 10% from the cooling flows; the remaining 40% is related to the exterior body (3). This is a very rough breakdown and can vary a lot depending on how the cooling drag and drag from wheels are defined; but still worth bearing in mind.

An underbody diffuser of a road-going vehicle, in racing applications, is used to generate down-force. For passenger vehicles there is a greater focus at drag reduction, and this function has been tested several times in the past. The function of the diffuser in automobile usage is complicated and dependent on many parameters.

Experiments with simplified bluff bodies in ground proximity were carried out by Cooper (4) and tested with different ground clearance, diffuser geometries and road boundary conditions. The test model was limited to only one type of upper body though, squareback. Down-force generation increases with increasing ground proximity until the ground clearance is small enough to make the boundary layer of the body interact with the ground surface. The down-force will then drastically drop. The trend with increasing ground proximity, for drag, was that it may reduce for large diffuser angles but remain unchanged for small diffuser angles.

Drag reduction relative diffuser angle varies with the upper body geometries has been shown in (5) (6). The underbody flow will interact with the flow of the upper body which will require optimization of upper and under-body together to reach optimum drag for the body. From tests with simplified bodies by P.Bearman et al. (7) it has been shown that a diffuser of 10 degrees, added to a fastback type of model, will reduce the drag significantly and the main contribution comes from a reduced vortex induced drag.

Undisturbed upstream flow is required to make a diffuser effective, which will make a combination of under-body covers with smooth transition to the diffuser a necessity.

Today many passenger cars use under-body covers giving a smooth underbody that by itself generally reduces drag, but diffusers are only present for sports/race cars, with very few exceptions.

The purpose was to extend the study (6) of a diffuser's effect to a sedan and wagon vehicle based on simulations in CFD. The focus was set on investigating the impact of underbody covers that include an angle at the rear end; making the rear end underbody act like a diffuser.

Automobile Diffuser

To install under-body panels and a diffuser to an existing car generally requires a change in the layout of the under-body. Components such as exhaust system, fuel tank, rear suspension and the body are placed as low as possible to maximize interior space of the vehicle. The ground clearance of a passenger car is generally as low as possible due to styling, handling and aerodynamic requirements and will not accept underbody covers without changes to existing components. There are many practical reasons for the chosen ground clearance requirements of a production car such as customer usability to clear speed bumps, car washes, garage ramps, truck ramps, pavements etc. Garage ramps and truck ramps normally require a certain ramp angle at the front and at the rear that may limit the feasibility of using spoilers at the front. At the rear this will limit the possibility to use guiding vanes or end plates for aerodynamic purposes. Passenger cars typically requires 14°-15° approach and departure angles at the gross vehicle weight ride height. To make as little change as possible to current vehicle concepts it is natural to locate the start of the diffuser in the region of the centre of rear wheel.

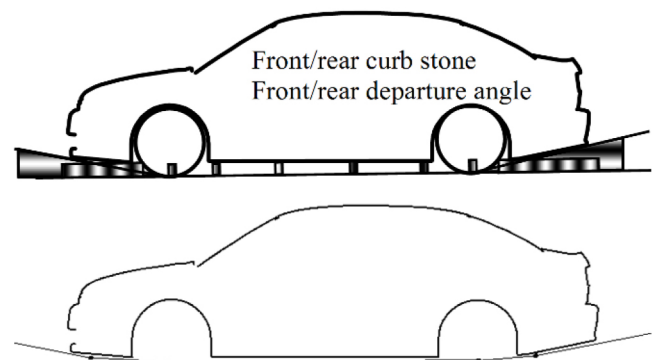


Figure 1. Ground clearance requirements of a Saab 9-3.

One issue arising with under-body covers is that an additional component is added to the vehicle, leading to extra mass, and an extra assembly operation in the production process. Benefits with under-body covers include reduced aerodynamic drag, possibly reduced need for wear protection and coating of the body, and sometimes less need of noise insulation. But, to fit the under-body covers it is best to integrate the components of the underbody in a way that it will create as smooth floor as possible. This is not feasible for

all configurations of rear suspension for instance, and may very well prevent the use of some types of rear axles. To cover the exhaust system will require a clearance envelope between the exhaust system and the underbody covers and this will lead to the fact that the exhaust has to be moved higher up in the tunnel, and perhaps thermal issues for other components may arise. It is generally not practical to fully cover the exhaust tunnel. The under-body shields require cut-outs for access to transport hooks and lifting jacks etc. The most difficult area to cover is the rear axle and the diffuser area, where the mufflers/silencers are located. To have a smooth transition from the underbody to the diffuser and maintain attached flow over the diffuser, is a requirement for achieving the desired function of the diffuser. Attached flow is necessary for pressure recovery over the diffuser. It will then have the function of directing the wake flow in an upward direction that will affect the camber of the flow around the body.

Diffuser Theory

An underbody designed as a convergent-divergent venturi channel with the road, generates a low pressure region on the under-body. As the air travels along the under-body, the flow will have a negative pressure peak where the diffuser starts. The air will accelerate at the diffuser inlet and decrease the pressure according to Bernoulli's law, which will generate down-force (8).

The complete function of an automobile diffuser is complex but the desire, when working with drag reduction, is to enable a pressure recovery over the length of the diffuser. The higher base pressure, the higher drag reduction. The fluid dynamic mechanisms working in a diffuser can be split up into three parts according to Cooper (4).

First, the ground interaction. In free stream flow a symmetrical body, with no angle of attack, will have no lift. When the body is moved closer to ground, the narrow section between the under-body and ground will accelerate the flow and create a lower pressure under the body that will create a force on the body directed toward the ground. The ground will generate a flow asymmetry around the body. The ground proximity will affect the flow in an inviscid and a viscous way. The inviscid effect will move the stagnation point with changed ground proximity and generate a flow asymmetry. Typically the stagnation point will move down when the body moves closer to ground. The viscous effects become significant when the boundary layer of the body interacts with the ground.

Second, the upsweep provided by the diffuser. The rear underbody will camber the body, resulting in a lower pressure on the under-body that will increase the down-force, or reduce the lift force. The upsweep can reduce drag without ground proximity as it may allow a pressure recovery and will be a key variable in this study.

Third, is the diffuser pumping. The cross-sectional area in a diffuser increases downstream and will result in a reduction of the air velocity over the length of the diffuser. If no separation occurs, the greater the cross-sectional area becomes, the greater the static pressure becomes. The pressure rise is a function of the ratio of the inlet and outlet area. These areas are dependent on the diffuser angle and the ride height of the vehicle. The outlet pressure of the diffuser is the base pressure of the car and since the base pressure is close to constant, most of the pressure recovery will be seen as a pressure decrease at the inlet. An under-body with diffuser will then have higher flow rate, and lower pressure, that will produce higher down-force. The diffuser can be considered to have pumped the flow of the under-body. The diffuser pumping effect will always increase the down-force and the magnitude is dependent on the ride height.

From experiment with simplified bluff bodies with static ground simulation (9) the effect from a diffuser on drag varied with the upper body. The reference was a model with a vertical backlight and zero diffuser angle but including a 15 degree taper in plan-view. Figure 2 shows the normalized drag of a bluff body and the influence of backlight angle. Four curves with different diffuser angles are plotted and all the curves achieve an optimum at approximately 15 degree backlight angle, but the magnitude of the improvement changed with diffuser angle. The wind tunnel tests were carried out on a small scale simplified model, with a Reynolds number of approximately 1 million and a ground clearance of 0.2 of the model height. The potential of a diffuser as a drag reduction device is dependent on the backlight angle.

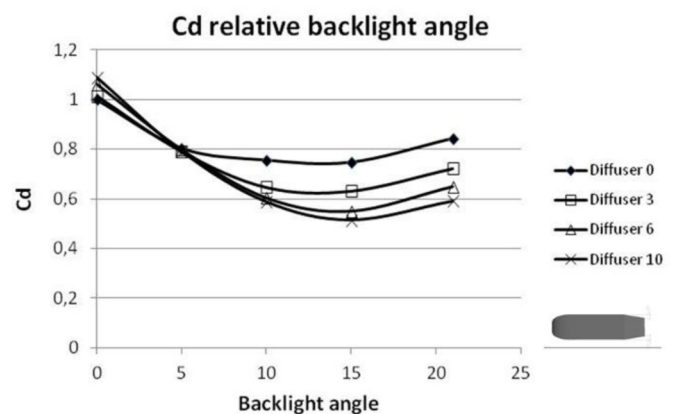


Figure 2. Effect of backlight and diffuser angle on a simplified, vehicle-like, bluff body in ground proximity, with static ground simulation. Cd was normalized with the drag of the reference model, with no diffuser angle or backlight angle.

This work was initiated to study this effect with a detailed passenger car model, under road-vehicle boundary conditions, by testing the effect of a diffuser to a sedan and wagon car.

METHOD

The work presented in this paper is the result of numerical computations in CFD, with the software Fluent. The models tested were identical at the front and underbody, with the only difference at the rear-end exterior shape, roof, sides and backlight/rear windscreen.

Test data was used to compare and tune the accuracy of the model: such as cooling flows and boundary conditions. The base cars, sedan and wagon, were tested in the Volvo wind tunnel to provide test data of the reference vehicles and tune the base model. The cars tested were a Saab 9-3 Sedan and Wagon of model year 2012. All the frontal areas used were reference frontal areas for the vehicles, that were standard at Saab. Reference area of the wagon was 0.05m^2 bigger than the sedan due to a higher roof-line. The underbody covers increased the front area by 0.0095m^2 , but this was not compensated for so the increased front areas are included in the coefficients. The cooling air flow was measured with an anemometer rack, using 12 anemometers between the charge-air cooler and the condenser in the cooling module. The measurements were carried out in the Saab Climatic wind-tunnel at a range of vehicle speeds from 40 to 200 km/h, to tune grille openings and cooling flows. The car tested was a diesel-engined variant with associated under hood and under-body shielding.

The Test Model in CFD

The numerical method was a standard automotive CFD technique. Surface mesh being a detailed geometry prepared in ANSA 13.2.3 using the same wheel geometry, ride height and outer geometry details as the tested cars. The mesh was prepared so that the only changes between the different set-ups were the under-body geometry with a variation of the diffuser angle. The resolution of the surface mesh was between 4 and 10mm to resolve suitable details and have required quality for the prism layer. The volume mesh was generated in Tgrid 14.0 with a prism layer of 15 layers and an aspect ratio of 5 from the surface mesh to maintain a y^+ of more than 30, as recommended by Fluent (10). The prism layer was applied to the exterior body and the smooth underbody covers. The volume mesh was a hexcore mesh with refinement zones for the under-body, wheels and wake region. An overview of the volume mesh can be seen in [Figure 3](#). The cooling module, condenser, charge-air cooler, radiator and fan shroud were simplified to one volume, and the mass flow was tuned by adjusting openings to the correct flow through the cooling pack and the engine bay. The simulation domain had a length of approximately 13 vehicle

lengths, a width of 11 vehicle widths and a height of 8 car heights, to remove the need for corrections for blockage. Boundary condition of the wind tunnel was a moving ground where the complete ground was moving. Walls and roof was set to symmetry. Final mesh size was approximately 65-70 million cells dominated by hexahedrals.

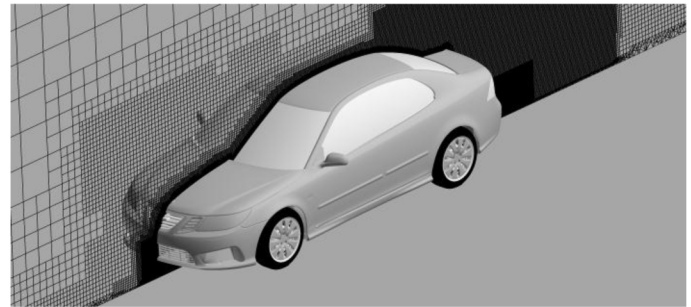


Figure 3. Overview of volume mesh showing refinement regions.

The solver used was Fluent 14.0. Simulations were carried out in RANS (Reynolds Averaged Navier Stokes) mode, pressure based with the k-epsilon realizable turbulence model. Non-equilibrium wall functions and a simple pressure-velocity coupling were used. 3000 iterations were performed for each case and it was considered converged when the variation was ± 1 drag count and the residuals were stable below 10^{-3} . The presented coefficients are an average from the last 200 iterations and the reference area was the same standard values as used in the tests. The wheel rotation was simulated with MRF (moving reference frame) and the wheel dimension and camber angles were the same as at the test car. Wheel geometry, such as radial and axial expansion, changed with wheel rotation speed, was not corrected for in this study, and the tread of the tyre was simplified as flat surfaces. The flow field was initialized with the fmg initialization process and the inlet boundary was set to 38,89m/s with turbulent intensity 1% and length scale of 0.1m. Outlet boundary was a pressure outlet with the same turbulence intensity and length scale for the reversed flow. The post processing was carried out in Fluent and FieldView 13. The geometry used in this work is shown in [Figure 4](#). The flat under-body plates cover the under-body from the engine bay. The engine is covered by the original engine cover and the front view reveals the floor is not fully in the same level from front to rear but close to.

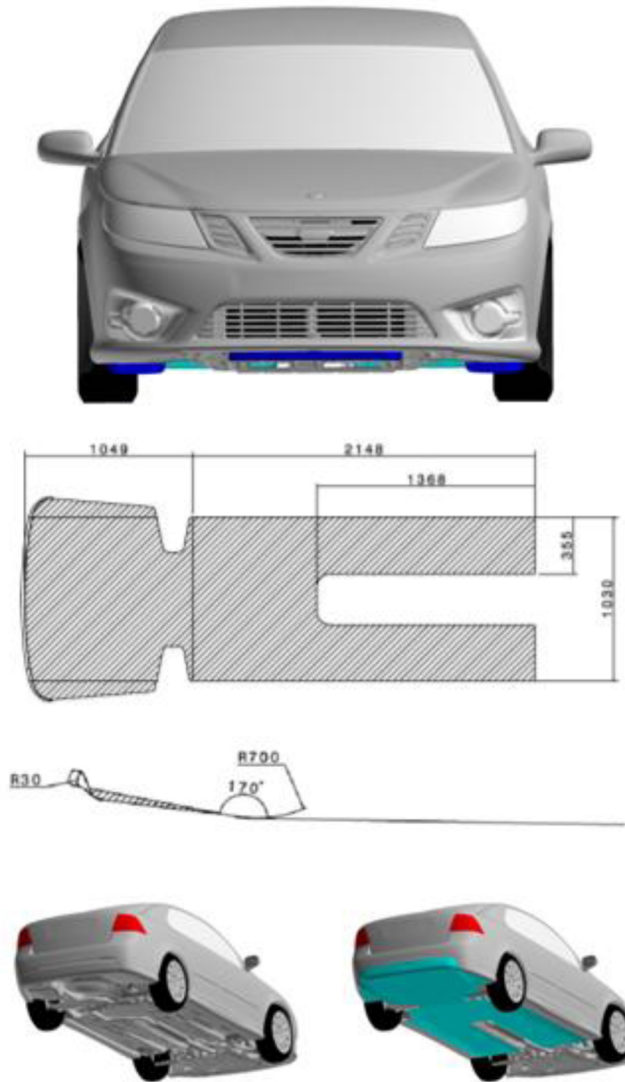


Figure 4. Geometry and description of the under-body covers and test vehicle.

Test Models in Wind Tunnel

The reference cars: sedan and wagon, were tested in the Volvo aerodynamic wind tunnel to tune the model as a reference. The simulated and tested results are compared initially. Cooling air flow has a big impact of the lift and drag of a vehicle. The air mass flow through the cooling module of the reference model was tuned to the same air flow as from the test of the reference car. The results are shown in Figure 5 and for the test speed 140 km/h this corresponded to approximately 1.25 kg/s. An error could also have been induced here due to measurement uncertainties and the fact that the measurement was only done to the wagon model. Simulations of the wheel rotation were carried out with MRF and the wheel-houses did not contain all details due to necessary simplifications.

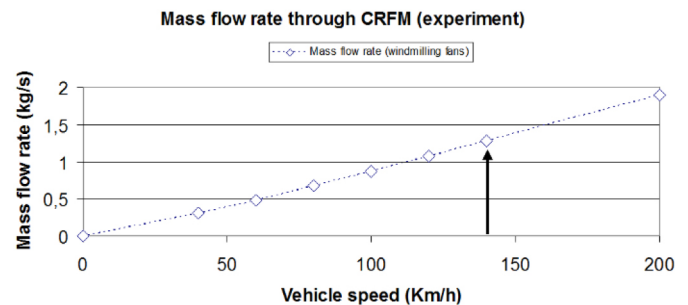


Figure 5. Mass flow rate through cooling module from test with full size car.

The comparison between simulated results and measurements are presented in Table 1. Results for the wagon car were generally under-predicted and results from the sedan were over-predicted. The drag was within 5% and considered to be acceptable, while the error was bigger for lift forces. Results from the measurements are corrected for horizontal buoyancy, solid model blockage and wake blockage according to Volvo method for slotted wall wind tunnel (11). The simulated result was uncorrected due to the over sized digital wind tunnel domain.

Table 1. Drag and lift of simulated reference models relative to results from wind tunnel measurement (Simulation -Test)

Configuration	Wagon	Sedan
ΔC_D	-0.013	0.008
ΔC_L	-0.067	0.034

Drag of the sedan is over predicted 8 drag counts in the simulations and under predicted -13 counts for the wagon. Lift is over predicted 34 counts for the sedan and under predicted -67 counts for the wagon. The drag correlated well but the discrepancies for lift were higher. This may be because the separation was not captured accurately in the simulation, and therefore produced an error in the surface pressures. Lift forces are more affected by boundary layer effects, especially prediction of the separation points. Friction forces do not apply to the lift forces but an error in the pressure forces above or under the car have a bigger effect to the plan-view area than the frontal area, simply because the frontal area is smaller. All details were not represented in the simulation model. There was also a difference in boundary condition between simulation and test. In the simulation the complete floor was moving and in the test only the areas that were covered by the five belt system was moving. The boundary layer in the wind tunnel was controlled by boundary layer suction and tangential blowing, which was unnecessary in the simulations. The engine bay was somewhat simplified in the simulation model and may have induced error in the pressure losses and how the air exited