

Adjoint-Driven Aerodynamic Shape Optimization Based on a Combination of Steady State and Transient Flow Solutions

Taeyoung Han, Shailendra Kaushik, and Kenneth Karbon
General Motors Co.

Benjamin Leroy, Kyle Mooney, Stamatina Petropoulou, and Jacques Papper
ICON Technology and Process Consulting Ltd.

ABSTRACT

Aerodynamic vehicle design improvements require flow simulation driven iterative shape changes. The 3-D flow field simulations (CFD analysis) are not explicitly descriptive in providing the direction for aerodynamic shape changes (reducing drag force or increasing the down-force). In recent times, aerodynamic shape optimization using the adjoint method has been gaining more attention in the automotive industry. The traditional DOE (Design of Experiment) optimization method based on the shape parameters requires a large number of CFD flow simulations for obtaining design sensitivities of these shape parameters. The large number of CFD flow simulations can be significantly reduced if the adjoint method is applied. The main purpose of the present study is to demonstrate and validate the adjoint method for vehicle aerodynamic shape improvements. Although steady-state Reynolds Averaged Navier Stokes (RANS) was used as a 'primal' solution for adjoint-based shape changes, a fully transient Detached Eddy Simulation (DES) was applied as the baseline and final flow solutions for improved flow accuracy. This type of analysis offers a more accurate flow modelling option, especially in cases where boundary layer separation occurs. The DES validation run was performed in the final stage of this study in order to confirm the drag coefficient reduction of $\Delta CD = -0.005$. The hardware validation results with a reduced scale clay model in a wind tunnel will be reported in future publications.

CITATION: Han, T., Kaushik, S., Karbon, K., Leroy, B. et al., "Adjoint-Driven Aerodynamic Shape Optimization Based on a Combination of Steady State and Transient Flow Solutions," *SAE Int. J. Passeng. Cars - Mech. Syst.* 9(2):2016, doi:10.4271/2016-01-1599.

INTRODUCTION

Computational Fluid Dynamics (CFD) has gained popularity as a tool for many airflow situations including road vehicle aerodynamics. This trend, to bring CFD to bear on vehicle aerodynamic design issues, is appropriate and timely in view of the increasing competitive and regulative pressures being faced by the automotive industry. Vehicle shape optimization for better aerodynamics performance is one of the most appealing ways of meeting these demands. The automatic optimization can be very beneficial in maintaining short vehicle production cycles. Conventional CFD flow information such as air velocity and pressure field do not provide direct guidance on the shape changes for improving the vehicle aerodynamic performance. This is because the 3-D flow field obtained from a standard CFD analysis is not explicitly descriptive to provide direct guidance on how changes to the vehicle shape impact the aerodynamic performance.

The adjoint method provides the design direction and the sensitivity information with only one solution of the adjoint equation, which is independent of the number of shape parameters. There is a long history of the use of the adjoint technique in CFD shape optimization with the major contributors being Jameson [1], Giles [2], Anderson [3] and others. More recent work from the authors on the adjoint

solver in topology and shape optimization can be found in [4], [5] and [6]. The process involves only one conventional CFD simulation and one adjoint solution in which the sensitivity of aerodynamic drag with respect to the shape is extracted at every surface mesh location on the vehicle. An optimized vehicle shape based on the drag sensitivities is then obtained. Although the adjoint method has been used extensively in many aerodynamic optimization problems, its application to full-scale vehicle design is still very challenging. The design engineers have to determine whether the suggested shape changes from the adjoint method are adequate to provide improved drag. At the same time, these changes must not conflict with other design constraints such as the interior volume, comfort, styling, various safety requirements and functionality.

The baseline vehicle shape was a production vehicle, already optimized using wind-tunnel testing and standard CFD processes at GM. The study is arranged in two phases. In phase 1, the optimization module was used to automatically loop through the primal-adjoint-morph steps, to reduce drag. In phase 2, a manual approach to optimization, especially developed for this investigation was adopted. In this approach, six potentially beneficial regions on the car surface (Figure 1) were identified using the adjoint sensitivity plots. From the pre-processing to the solver stages, every operation

described in this paper has been completed using iconCFD®[7] software developed by ICON Technology and Process Consulting Ltd®. These were sequentially morphed, one zone at a time, using the box-morphing toolset available in ANSA® [8]. This enabled larger surface deformations while maintaining smoothness and integrity of the morphed surfaces in the identified regions which was not possible in phase 1 approach.

ADJOINT EQUATIONS

The adjoint method is used for the calculation of the shape sensitivities of the vehicle. The adjoint equations implemented in the software are given in [9]-[10]. The required source terms and the adjoint boundary conditions are formulated as functions of the fluid dynamic objectives. The main equations of [10] are briefly outlined here. In this case, the porosity term is omitted from the adjoint momentum equation since the interest is in the surface sensitivities and no topology optimization is taking place. Assuming a generalized cost function, J , on the computational domain, Ω :

$$J = J(\mathbf{v}, p) = \int_{\Gamma} J_{\Gamma} d\Gamma + \int_{\Omega} J_{\Omega} d\Omega, \quad \Gamma = \partial\Omega \text{ or part of } \partial\Omega$$

Where J_{Γ} is the surface contribution of the cost function and J_{Ω} is the volume part. $\mathbf{v}$ is the primal velocities and p is the primal pressure field.

- The steady-state, incompressible Navier-Stokes adjoint equations with frozen turbulence takes the form:

$$\begin{aligned} -2D(\mathbf{u})\mathbf{v} &= -\nabla q + \nabla \cdot (2\nu D(\mathbf{u})) - \frac{\partial J_{\Omega}}{\partial \mathbf{v}} \\ \nabla \cdot \mathbf{u} &= \frac{\partial J_{\Omega}}{\partial p} \end{aligned}$$

$$\text{where } 2D(\mathbf{u})\mathbf{v} = -\nabla \mathbf{u} \cdot \mathbf{v} - (\mathbf{v} \cdot \nabla) \mathbf{u}$$

- The adjoint boundary conditions for walls and inlets are:

$$\begin{aligned} \mathbf{u}_t &= 0 \\ \mathbf{u}_n &= -\frac{\partial J_{\Gamma}}{\partial p} \end{aligned}$$

- The adjoint boundary conditions for outlets are:

$$\begin{aligned} q &= \mathbf{u} \cdot \mathbf{v} + \mathbf{u}_n \mathbf{v}_n + \nu (\mathbf{n} \cdot \nabla) \mathbf{u}_n + \frac{\partial J_{\Gamma}}{\partial \mathbf{v}_n} \\ 0 &= \mathbf{u} \cdot \mathbf{v} + \mathbf{u}_n \mathbf{v}_n + \nu (\mathbf{n} \cdot \nabla) \mathbf{u}_t + \frac{\partial J_{\Gamma}}{\partial \mathbf{v}_t} \end{aligned}$$

- The sensitivity derivatives with respect to an outward movement, β , on the surface have the following form with A being the surface area affected by this movement:

$$\frac{\partial L}{\partial \beta} = -A\nu (\mathbf{n} \cdot \nabla) \mathbf{u}_t \cdot (\mathbf{n} \cdot \nabla) \mathbf{v}_t$$

$$\text{where } L = J + \int_{\Omega} (\mathbf{u}, q) R d\Omega,$$

R stands for the residuals of the Navier-Stokes equations, $\mathbf{n}$ is the unit normal vector, $\mathbf{v}$ and $\mathbf{u}$ are the primal and adjoint velocities respectively, p and q are the primal and adjoint pressures respectively and ν is the kinematic viscosity. The subscript n and t represent the normal and tangential component of a field respectively.

The cost function used for the measurement of the drag forces reduction was:

$$J = \int_S \mathbf{d} \cdot \left(\mathbf{n}p - \nu \frac{d\mathbf{v}}{\delta n} \right) ds, \quad \begin{cases} d = (1, 0, 0) \text{ for drag} \\ d = (0, 0, 1) \text{ for lift} \end{cases}$$

where S is usually the car surface. The definition of the cost function is such that only the pressure component of the drag is taken into account for the optimization. Since the pressure component is dominant, this definition is adequate.

SOLUTION METHODOLOGY

A spreadsheet based preprocessing tool (iconCFD MultiCase©) was used to quickly prepare a number of simulation cases, ready for submission using STL geometry assemblies and user-defined templates (e.g. wind tunnel for automotive applications). The optimization process was divided into two phases:

Phase 1

In the first phase, the exterior of the vehicle was morphed through an iterative optimization process. Some geometry preparations were initially required to group geometry patches together in order to control morphing and the drag coefficient improvements (monitored between each cycle). The optimization consisted of a RANS flow run also called the “primal” run, an adjoint run where the sensitivities were computed, and a morphing stage. The direct deformation morpher works as follows: the sensitivities are translated into point displacement and scaled back to restrict the deformation to a given maximum displacement, followed by further smoothing. The morphing is applied on the geometry triangulated grid and exported in STL format in order to be used in the subsequent optimization steps. The normalized sensitivities (scalar values) are used in the morphing, and the local surface displacements are proportional to the local sensitivity vectors (“push in” or “pull out”). During the first phase, the shape sensitivities were applied to the entire vehicle surface and the total drag improvement due to the shape change evaluated. The maximum allowed displacement is defined by the user and the effects of the shape change on the total drag coefficients is investigated.

Phase 2

In the second phase, the exterior surfaces that demonstrated the highest sensitivities were selected for individual sub-region optimization, as shown in [Figure 1](#). The optimization process exploited a “front-to-rear” approach (the direction in which the morphing takes place), where each subsequent run used the previous sub-section geometry modification. A flow simulation followed by an adjoint sensitivity simulation was then performed for each configuration.

During the second phase, surface morphing was applied using the CAD pre-processing tool in which morphing boxes were created around each sub-region. The morphed geometry from the first phase

was used as a guide for the individual sub-region morphing. The morphing boxes were introduced in such a way that the symmetry of the end design was preserved (see [Figure 2](#)).

Certain geometric constraints were introduced during the CAD software morphing process; for example, the mirror glass surface was unchanged (see [Figure 3](#)).

For each sub-region morphing, the drag improvements were recorded by RANS simulations. Although RANS simulations tend to predict the drag increments due to shape changes, DES simulations are considered more accurate for the overall drag predictions. Therefore, in order to confirm the total drag reduction, DES confirmation runs were performed on the original and fully morphed designs after the completion of the second phase.

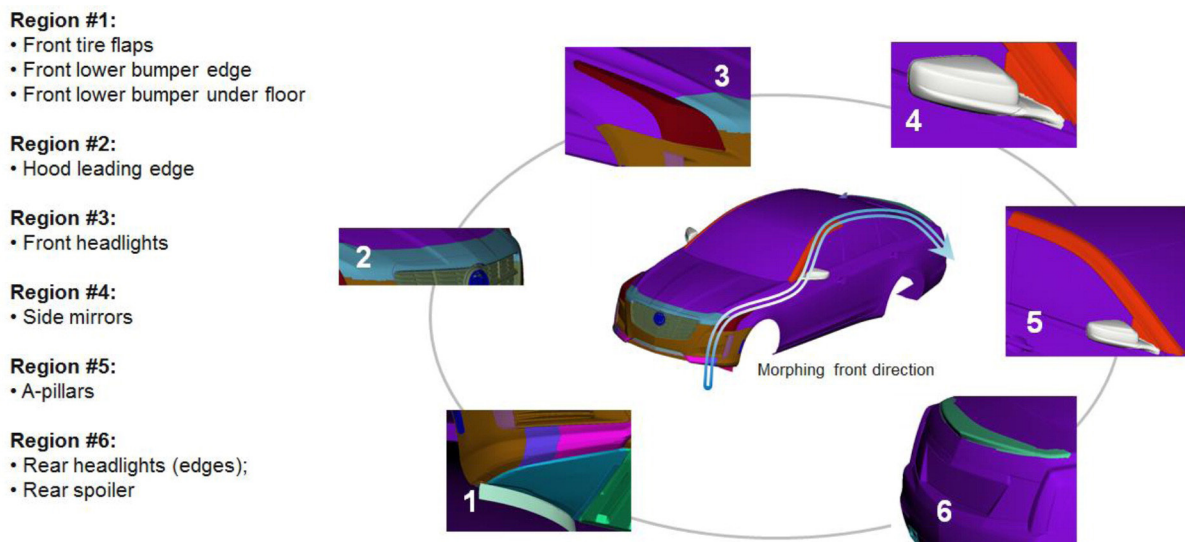


Figure 1. Selected areas for local optimization

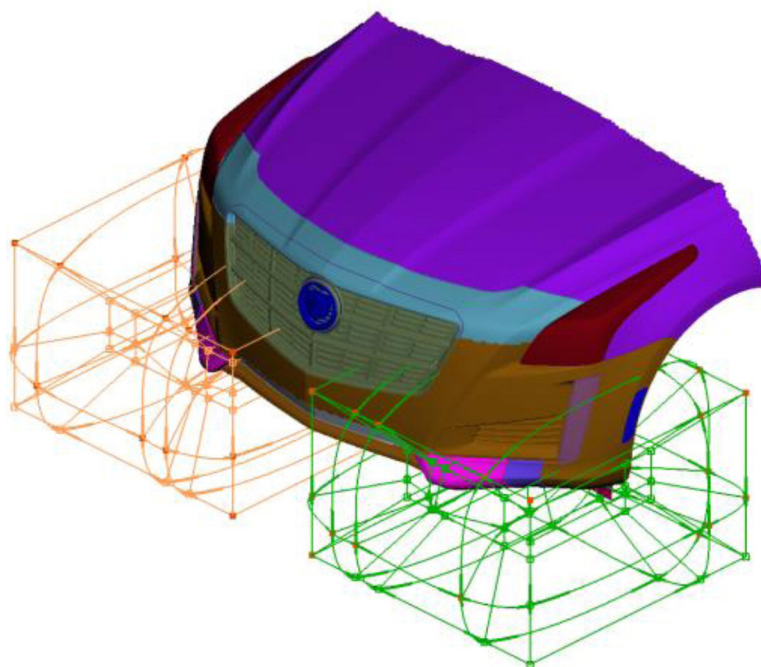


Figure 2. Example of the morphing boxes around the front bumper.

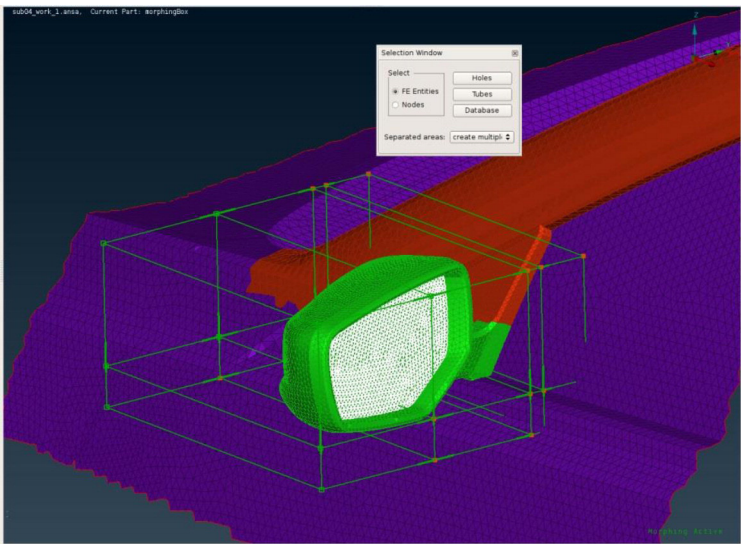


Figure 3. Introducing morphing constraints through the CAD software. Constrained mirror surface.

RESULTS

All cases presented in this study were completed on a GM production vehicle as seen in Figure 4. As the focus is on external aerodynamic changes, the under-hood compartment was closed with a surface on the front grille. For the computational domain, a standard wind tunnel model as represented in Figure 5 was used. The main boundary conditions are presented in Table 1.

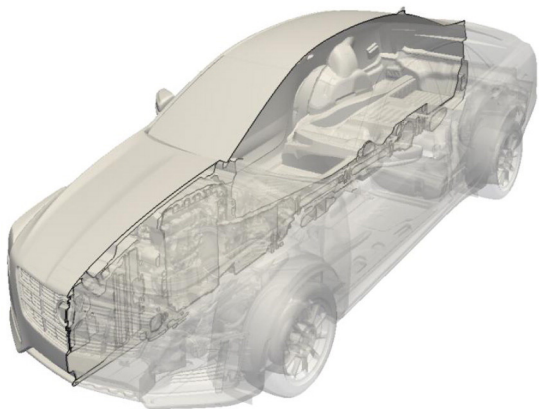


Figure 4. Car CAD model with closed under-hood compartment.

Figure 6 shows the vehicle geometry and the velocity field in the symmetry plane. A predominantly polyhedral mesh with prism layers at the walls has been used in the primal flow and the adjoint simulations. Wall Y+ distribution on the main external aerodynamic surfaces of the car are within a range of [30-40]. In the second phase, more detailed analysis was required due to the local areas of interest, so further refinement of the surface and overall mesh was applied in phase 2. This led to a mesh size (in the second phase) to approximately 85M cells compared to 26M cells in the first phase. A RANS, steady state solver with the Realizable k-ε model [11] was used for the drag calculations between optimization loops.

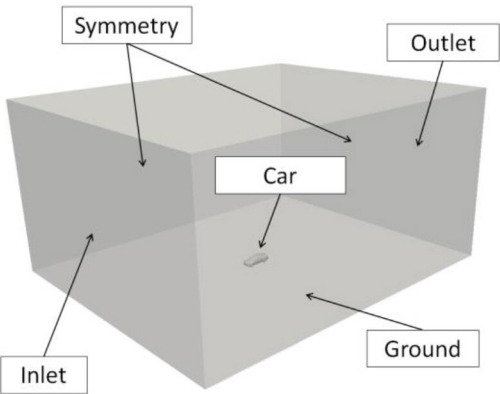


Figure 5. Wind tunnel setup

Table 1. Boundary conditions description

Part ID	Boundary condition types
Inlet	Velocity inlet (30m/s)
Outlet	Pressure outlet
Ground	Static wall
Car body	Static wall
Car wheels	Static wall

For phase 1, the whole exterior styling was modified during three consecutive optimization iterations. The drag coefficients as well as the changes in frontal area can be seen in Table 2. For this phase the process was fully automated. Simulations were considered converged when the drag coefficient oscillations reached an amplitude of 0.5drag count.

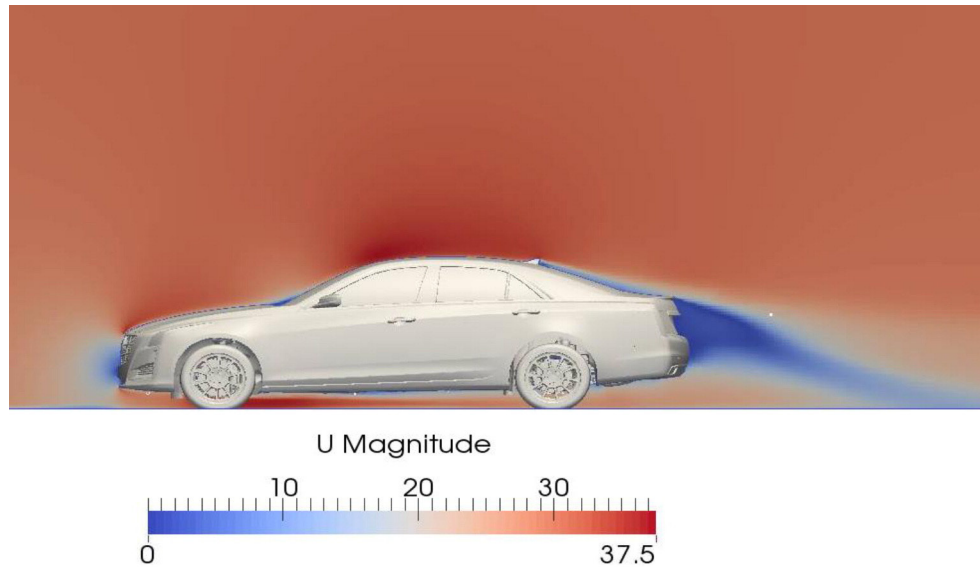
Figure 6. RANS solver velocity field $y=0$ cut.

Table 2. Phase 1 - Drag coefficient reduction during iterations.

Case ID	Area modified	ΔC_d (1 count=0.001)	Frontal Area (m^2)
Original (Baseline)	Base	-	2.321
Morphed 1	Exterior styling	-3.6	2.312
Morphed 2	Exterior styling	-3.8	2.305
Morphed 3	Exterior styling	-3.5	2.299

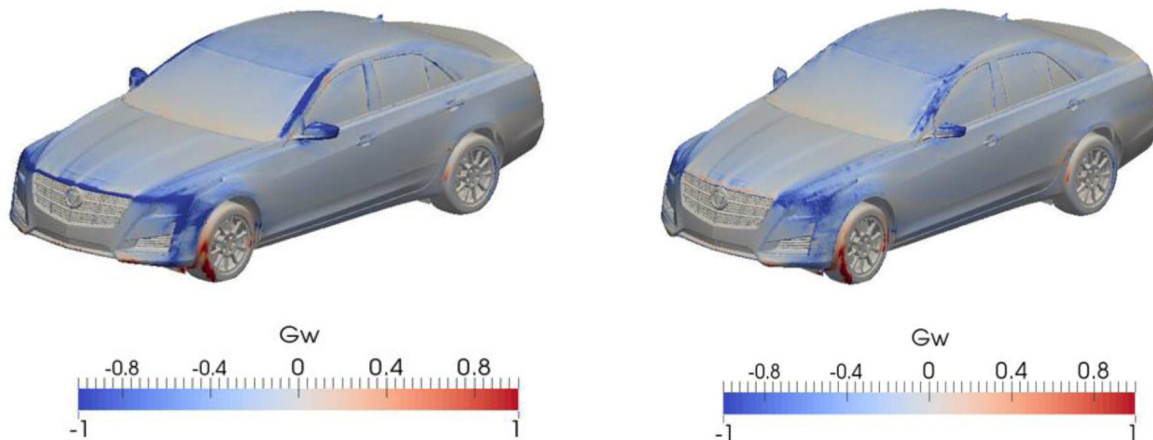


Figure 7. Phase 1 - Comparison of original (left) and final (right) sensitivities.

As the geometries was morphed, the frontal area changed (see [Table 2](#) and [Table 4](#)) and in order to consistently compare the drag coefficients a constant reference frontal area of $2.321m^2$ was used (original geometry frontal area value).

The surface sensitivities of the baseline versus the last design can be seen in [Figure 7](#). The strength of the shape sensitivity was softened from the baseline shape to the final morphed shape. A total drag reduction of 3.8 counts was achieved in this phase. The maximum displacement was limited to 20 mm for each optimization iteration and to 30 mm for the overall optimization process. The areas that reached the overall maximum displacement of 30 mm were excluded from the next optimization iteration. The maximum of 30 mm was achieved in some of the most deformed areas. The relatively small