

Aerodynamic Shape Optimization of an SUV in Early Development Stage using a Response Surface Method

Shaoyun Sun
FAW R&D Center

Yin-ping Chang
Oakland University

Qiang Fu, Jing Zhao, Long Ma, and Shijie Fan
FAW R&D Center

Bo Li, Andrea Shestopalov, Paul Stewart, and Heinz Friz
Exa Corporation

ABSTRACT

In the development of an FAW SUV, one of the goals is to achieve a state of the art drag level. In order to achieve such an aggressive target, feedback from aerodynamics has to be included in the early stage of the design decision process. The aerodynamic performance evaluation and improvement is mostly based on CFD simulation in combination with some wind tunnel testing for verification of the simulation results. As a first step in this process, a fully detailed simulation model is built. The styling surface is combined with engine room and underbody detailed geometry from a similar size existing vehicle. From a detailed analysis of the flow field potential areas for improvement are identified and five design parameters for modifying overall shape features of the upper body are derived. In a second step, a response surface method involving design of experiments and adaptive sampling techniques are applied for characterizing the effects of the design changes. The characterization is followed by an optimization step to find the best possible drag improvement from these design changes. At the same time the wind tunnel tests at the Shanghai Automotive Wind Tunnel Center with a fully detailed clay model of the vehicle confirmed the performance improvement of the simulation results.

CITATION: Sun, S., Chang, Y., Fu, Q., Zhao, J. et al., "Aerodynamic Shape Optimization of an SUV in early Development Stage using a Response Surface Method," *SAE Int. J. Passeng. Cars - Mech. Syst.* 7(4):2014, doi:10.4271/2014-01-2445.

INTRODUCTION

In today's economic environment, consumer demand for fuel-efficient vehicles has never been higher. Vehicle aerodynamic design has a critical impact on fuel efficiency, through reducing wind resistance of the vehicle's exterior shape and reducing losses associated with requirements for cooling flow through the engine compartment, which has been summarized by Hucho [1]. Aerodynamic design starts with the earliest concepts of the vehicle based on its shape and proportion to meet styling intent, passenger space, and component packaging needs. As the exterior shape is refined, the aerodynamic efficiency is driven by shape parameters such

as angles, radii, and dimensions. Typically, improvements can be made in these parameters with minimal impact on the styling aesthetics. The reference [2] has described the general role of aerodynamics in the whole process of vehicle development.

The challenge faced by vehicle manufacturers in each design stage is the urgent need for information about how to improve the design. Aerodynamic information can be costly and difficult to obtain, traditionally involving building a detailed model or prototype of the vehicle and testing the model in a wind tunnel. Design iterations at this late stage of product development are time-consuming and costly because it is difficult to make

large-scale changes to a model, or to change any surface features of a fully detailed prototype during a wind tunnel test. Prototype testing for aerodynamics is a major contributor to vehicle development costs and design cycle time.

Aerodynamics simulation changes the vehicle development process, reducing both vehicle development costs and design cycle time. Because of its inherent advantages over physical testing methods, simulation can bring much more feedback about the design performance into each stage of development, improving the ability for designers and engineers to innovate in balancing design aesthetics with aerodynamics.

In the early development stage of an FAW's sports utility vehicle (SUV), the upper body is optimized for best aerodynamic performance. At first, the performance of current design is evaluated by computational fluid dynamics (CFD) simulation. Based on simulation results, the potential areas of improvement are identified. The aerodynamics team then discussed with the styling team to determine the range of design changes so that the design space is defined. A total of five parameters, each parameter represents a design change of the shape of the upper body in a certain range, are defined to form the design space. A minimal Latin Hypercube Sampling (LHS) [3] method plus an adaptive sampling method are used to generate the run list of simulations used to quantify the design space and corresponding response space. A response surface method is used to both characterize the effects of the design space on the performance objective and search for predicted optimal design regions within the design space. The optimum design calculated by the response surface is validated by CFD simulation.

Simultaneously, an open-grille clay model of the current SUV (baseline geometry) is built and experimentally tested in a wind tunnel. The clay model is later modified to test the performance of the optimized shape. The wind speed and the scales of the models are set to be the same in simulation model and in wind tunnel test model.

METHODOLOGY

Simulation Method

In the early development stage of the SUV, one version of the CAS (Computer Aided Styling) surface is used as the basis of the aerodynamic optimization work. The CAS surface is then cleaned and combined with the underhood and underbody geometry of an existing vehicle so that a fully detailed digital model is built and then used for analysis.

Commercial software PowerFLOW is used to perform transient (unsteady) flow simulation. This software uses a special discretization of the Lattice Boltzmann method (LBM) on a variable resolution Cartesian volumetric grid, which is automatically generated by the software. The key concept of the LBM is very briefly outlined here. For more details, please refer to literature [4,5,6,7,8].

The Boltzmann equation describes a representation of the particles and how the particles evolve in a fluid. The equation can be written as following,

$$\frac{\partial f}{\partial t} + \vec{v} \cdot \frac{\partial f}{\partial \vec{x}} = \Theta \quad (1)$$

where f is the velocity probability distribution function and Θ is the collision operator, which satisfies the necessary conservation laws. v is velocity vector, t is time and x is the position vector.

In the lattice form, the Boltzmann equation can be expressed as a set of algebraic equations of the probability distribution function at each state, f_i ,

$$f_i(t + \Delta t, \vec{x} + \vec{e}_i \Delta t) = f_i(t, \vec{x}) + \Theta_i(t, \vec{x}) \quad (2)$$

In which e is the unit normal vector. Here the lattice BGK collision term is,

$$\Theta_i(t, \vec{x}) = -\frac{\Delta t}{\tau} [f_i(t, \vec{x}) - f_i^{eq}(t, \vec{x})] \quad (3)$$

where τ is the relaxation time and f_i^{eq} is the lattice equilibrium distribution function.

The macroscopic variables such as density ρ and velocity v can be expressed by the distribution function as following,

$$\rho(t, \vec{x}) = \sum_i f_i(t, \vec{x}) \quad (4)$$

$$v(t, \vec{x}) = \frac{1}{\rho(t, \vec{x})} \sum_i f_i(t, \vec{x}) \vec{e}_i \quad (5)$$

Turbulence modeling in this study uses turbulence theory to model only the dissipative and inertial ranges, modeling the dynamics of sub-grid scale turbulence by two additional equations derived from an extended RNG (Renormalization Group theory) formulation, as following,

$$\rho \frac{Dk}{Dt} = \frac{1}{\sigma} \frac{\partial}{\partial x} \left[(\mu + \mu_t) \frac{\partial k}{\partial x} \right] + P - \rho \varepsilon \quad (6)$$

Here k is turbulence kinetic energy, μ is dynamic viscous coefficient, μ_t is turbulence viscous coefficient, P is pressure and ε is the dissipation of the turbulence kinetic energy.

For more details, please refer to literature [9] and [10]. This approach is also known as Very Large Eddy Simulation (or VLES).

The aerodynamic CFD simulations were performed in the Digital Wind Tunnel (DWT). DWT is a case template for simulating the vehicles in open road condition. The whole floor of the simulation is treated as sliding wall boundary condition with the same velocity of the ambient flow in order to represent real world driving condition. The thickness of the boundary layer is zero in front of the vehicle. The wheels are all treated as rotating wall boundary conditions in all simulation cases.

In the simulation model the full scaled vehicle is used. The best practice guide [11] for aerodynamic simulation of SUV is used to setup the case and the variable resolution regions. The total voxels (volumetric grids) of the baseline case is about 146 million. For the physical time scaling, the time step is 6.683×10^{-6} second. The total time steps are set to be 300,000 (about 2 seconds of real time). For each simulation, backward averaging technique is used to ensure the convergence of transient simulation. The convergence criteria in all the simulation is set to be one count for the drag coefficient.

For the simulation boundary conditions, the ambient air speed is set to be 120 km/h to match the speed limit on typical Chinese highways. The Reynolds number in this case is 3.54×10^6 , which is the same as in the wind tunnel. There is a fixed turbulent intensity (0.01) and a fixed turbulent length scale (5mm) in the simulation inlet boundary condition to estimate the turbulent state in reality. With these conditions, the boundary layers on vehicle surface are assumed as fully developed turbulent boundary layers. There is no measurement of the laminar-turbulent transition locations on the vehicle surface in the wind tunnel tests. The heat exchangers in the underhood are simulated as porous media and the pressure drop coefficients are calculated by experimental data of the components from the existing vehicle.

Optimization Method

There are examples of advanced design methods for optimization, such as Response Surface Methods and Design of Experiments, of use in the automotive field [12]. However, it has not been our experience that these methods are ubiquitous in automotive design. Historically the more common method, especially in cases involving complex flow physics or geometrical changes that are not well understood, is hypothesis testing [13,14,15].

Hypothesis testing begins by performing one or more simulations on baseline geometry. From the results one can reason about the behavior of the fluid system and by extension reason about likely performance changes that might be achieved by changing aspects of the baseline geometry. The engineer's hypotheses are translated into geometric design

variations to be tested. The results of each test are used to guide the next variant, or if the tests were done in parallel the best performing configuration is selected as the winner. The serial mode of this design methodology has a certain utility in specific environments, such as wind tunnel testing of a clay model where variations can be time consuming to create and even more difficult to repeat. The parallel mode is also still popular in automotive design because it allows for infinite flexibility in terms of design parameters, while allowing for the engineer to limit the number of variants tested. Often hypothesis testing is used when time or resources are limited and more sophisticated statistical methods are judged to be too costly.

The primary inefficiency of hypothesis testing, however, is that data from each variant is only valid for that variant. Knowledge gained from this variant is only used to drive the following variant. If the production problem of interest contains many design parameters with interactive effects on performance, the influence or importance of a single parameter may not be reliably deduced from any one model.

Advanced methods that can overcome this inefficiency include, but are not limited to, design using a response surface model. Response surface models can be used as a representative or surrogate of the true response of CFD simulations. In this way, interrogation methods such as analysis of trends, the main and interaction effects of parameters, and global search of optima, can be performed at little or no cost once the model is built. Additionally, multiple response models can be built from the same input data providing an analytical way to relate multiple objectives. This additional information extracted from the data set via the response surface model gives each individual simulation more and lasting value to the process.

The barrier to response surface methods being employed more widely is the cost of constructing the data set. For simple linear trends, one need only evaluate simulations equal to 2 times the number of input parameters. However for more complex systems, the general rule in generating data via Design of Experiments techniques is 8-10 times the number of input parameters. For the method described here, we are using adaptive sampling (AS) employing a variant of the weighted expected improvement function (WEIF) described by Sobester, et al, in [16], followed by iterative validation of optima to improve local convergence. In practice we have seen this method require simulations totaling 3-5 times the number of input parameters.

When fully realized, response surface methods allows the engineer to focus more on creating parameters that will have significant effects on performance while maintaining the design character of the vehicle. The system is then allowed to perform the challenging tasks of finding complex trends, interactions and optima.

Response surface models take as input data from simulations in the form of independent and dependent variables. We refer to the independent variables as the design space (DS) parameters and to the dependent variables as the response space (RS) parameters. As an example, a design space could consist of geometric morphs of a car's front fascia, rear spoiler, and rear diffuser. An example of a response space could then be the car's drag coefficient, as well as front and rear lift coefficients. The response surface model constructs a mathematical representation of a response given the matrix of design space parameters and a vector of responses for a series of simulations. A widely used type of response surface model is the Kriging surface [17]. The Kriging surface assumes trends are characterized by the following equation,

$$\hat{Y} = f(X)\beta + e \quad (7)$$

Where X is the input matrix of DS parameter values for a number of simulations, $\hat{Y}$ is the predicted response, β are coefficients to a polynomial regression, f , of order selected by the user, and e is "white noise" which is solved for via a user-specified correlation function and the constraint that the response must exactly pass through the available data.

Once the response surface model is constructed, a variety of analysis techniques can be used to extract information from the data set.

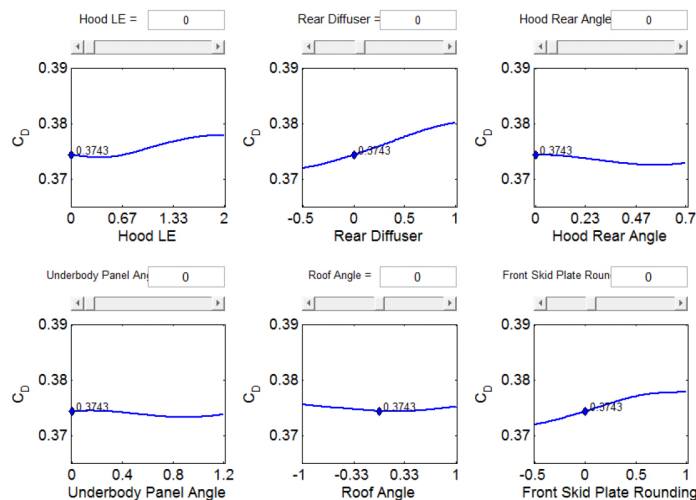


Figure 1. 1-D visualization of response surface model

The first level of analysis is simply visualization of the response surface itself. This can be done by viewing the trends in one parameter direction at a time so the response surface appears as a line or in two parameter directions such that the response is a visible surface. When the response surface model has many dimensions, the one parameter, or 1-D, visualization, as

shown in Fig. 1, is useful to simultaneously view trends in many dimensions from a single location in the design space. The two-parameter, or 2-D, visualizations, as shown in Fig. 2, help predict interaction effects between parameters at a single location in the design space. More complex analysis of the response surface can be done by averaging parameter main effects [18] over the entire design space to return which parameters have the largest effect on the response. Finally the response surface can be directly searched to find the predicted optimum configuration, or in the case of multiple responses, to create a predicted Pareto front as seen in Fig. 3, in which the multiple response of the drag coefficient C_D and the front lift coefficient C_{L_f} are shown. This search can also involve restricting the design space to let the designer explore "what if" scenarios.

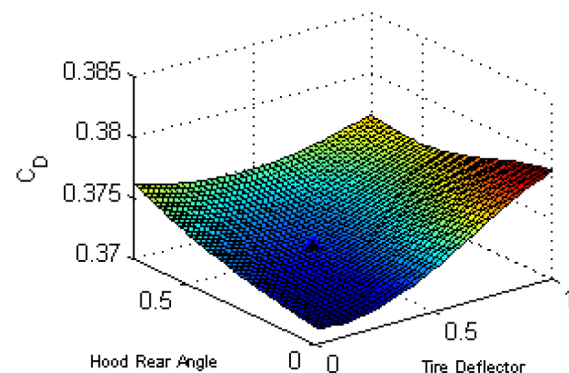


Figure 2. 2-D visualization of response surface model

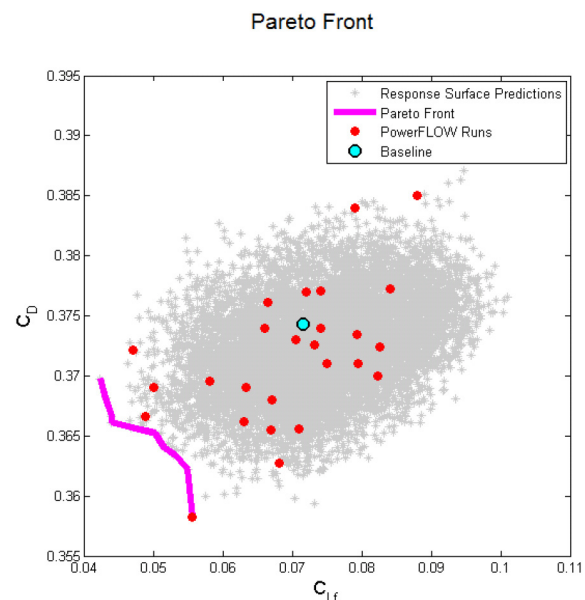


Figure 3. Pareto Front prediction via response surface model