

Automated Aerodynamic Vehicle Shape Optimization Using Neural Networks and Evolutionary Optimization

Anton Lundberg, Per Hamlin, and Davangere Shankar
ÅF Industry

Alexander Broniewicz, Tim Walker, and Christoffer Landström
Volvo Car Corp.

ABSTRACT

The foremost aim of the work presented in this paper is to improve fuel economy and decrease CO₂ emissions by reducing the aerodynamic drag of passenger vehicles. In vehicle development, computer aided engineering (CAE) methods have become a development driver tool rather than a design assessment tool. Exploring and developing the capabilities of current CAE tools is therefore of great importance.

An efficient method for vehicle shape optimization has been developed using recent years' advancements in neural networks and evolutionary optimization. The proposed method requires the definition of design variables as the only manual work. The optimization is performed on a solver approximation instead of the real solver, which considerably reduces computation time. A database is generated from simulations of sampled configurations within the pre-defined design space. The database is used to train an artificial neural network which acts as an approximation to the simulations. Finally an optimal vehicle shape is determined using the particle swarm optimization method. The method is solver independent and can handle multiple objectives.

The method was incorporated in an optimization tool compatible with Volvo Car Corporation's aerodynamics computational fluid dynamics (CFD) process. The capabilities of the optimization tool were demonstrated on a simplified low-drag car model. An improved shape with a 13.0% lower C_D was achieved with a prediction error of 0.4%.

CITATION: Lundberg, A., Hamlin, P., Shankar, D., Broniewicz, A. et al., "Automated Aerodynamic Vehicle Shape Optimization Using Neural Networks and Evolutionary Optimization," *SAE Int. J. Passeng. Cars - Mech. Syst.* 8(1):2015, doi:10.4271/2015-01-1548.

INTRODUCTION

The concentration of greenhouse gases (GHG) in the atmosphere is the highest it has been in the last 800,000 years [1]. The consequences of further increased atmospheric GHG levels are potentially very harmful. To battle climate change the European Union (EU) issued a climate policy called the "20-20-20" targets in 2009. The main objective of this policy is to reduce the emissions of GHG by 20% in 2020 compared to 1990 levels [2].

Data from 2010 shows that the transport sector accounts for 31.7% of the entire energy consumption in the European Union [3]. Part of the EU policy is a limit of CO₂ emissions from passenger cars, which for 2020 is set to 95 g/km with heavy penalties for excess emissions [4].

Improving the aerodynamic shape of a vehicle is an important factor in reaching the above mentioned limits for car manufacturers. The aerodynamic drag greatly exceeds 50% of the total driving resistance at highway speeds [5]. Hence, reducing the drag can induce significant improvements to fuel economy and CO₂ emissions.

Computer aided engineering (CAE) is today a vital part of passenger vehicle development and design process. It has become a development driver tool rather than a design assessment tool. Present day vehicle aerodynamics development is heavily dependent on computational fluid dynamics (CFD). Exploring and developing the capabilities of current CFD tools is therefore of great importance. Typically, the aerodynamic drag of a model is improved by analyzing flow simulations individually and manually updating the model. This is a costly, time-consuming and labor-intensive process. Automated optimization can help make the CFD development process more labor and cost efficient.

The aerodynamic behavior of passenger vehicles is seldom completely linear. Not only can the main effects of specific changes to the geometry be non-linear in themselves, interference effects between different parts of the vehicle are frequent. The response when conducting parameter studies is therefore very difficult to predict. An optimization tool that can incorporate non-linearity and maintain its prediction ability is therefore required in order to properly treat the complex optimization problem at hand.

The presented work is part of a larger project conducted at Volvo Car Corporation (VCC) in collaboration with ÅF Industry. The project aims at developing conceptual aerodynamics solutions within the environmental and legislative context described above. Automated optimization is an immense asset that allows efficient concept development in the project.

DEVELOPMENT OF OPTIMIZATION TOOL

Framework

The schematic of a typical method for optimizing the aerodynamic shape of an object is depicted in Figure 1.

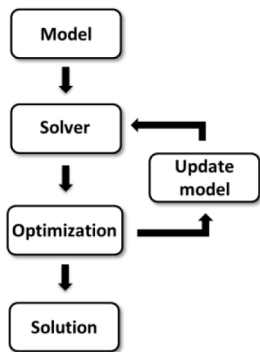


Figure 1. Schematic of a general method for vehicle shape optimization.

The method starts with a single initial model. A CFD simulation is performed for this model, whereby the result is analyzed using a certain criteria or optimization algorithm. An arbitrary number of model design parameters are updated whereby a new CFD simulation is performed. The iterative process continues until a stop condition is reached. This method is successfully demonstrated in [6,7,8]. The approach is however computationally demanding and there is a large potential for improvements.

One way to reduce the computational cost is to perform the optimization using an approximation of the solver, an approach often referred to as *two-level optimization*. The use of such a procedure can considerably decrease the computation time [9]. A schematic flowchart of this method is depicted in Figure 2.

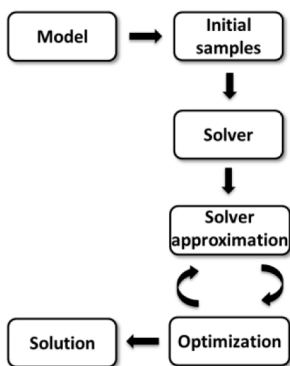


Figure 2. Schematics of two-level vehicle shape optimization.

Instead of performing a new CFD simulation every time the model is updated, initial simulations are performed on a limited number of samples within the pre-defined design space. The samples are chosen using a design of experiments (DoE) method, yielding a statistically optimal representation of the design space. The database of results from these initial simulations is used to create an approximation model reflecting main and interaction effects of the design variables. The actual optimization is subsequently performed on the approximation model. This approach was successfully adopted in [10,11]. There are a number of method choices involved in developing such a workflow, including: solver method, design space definition, the choice of a design of experiments method, a solver approximation method and an optimization algorithm.

Solver

The developed optimization method is solver independent and can therefore be used together with any CFD solver. The focus of this study is on the optimization method whereas any evaluation or improvement of the CFD process is not performed. In this particular application, the flow is solved according to CFD guidelines available in-house at VCC. A short description of the steps involved in this process is given below:

1. Computer aided design (CAD) model treatment and surface meshing in ANSA.
2. Volume meshing in HARPOON. Typically generating a mesh consisting of 10 million to 100 million cells, depending on the configuration.
3. Steady-state flow field solved in ANSYS Fluent with inlet air velocity set to 100 km/h, using the realizable k- ϵ turbulence model, and applying standard wall functions.
4. Post-processing in EnSight, Python and Excel.

During previously conducted studies at VCC including [12], this process has been observed to reasonably well predict changes to C_D and C_L when comparing to wind tunnel measurements. Therefore, the general trends in the CFD simulations can be expected to be observed also in the real world.

Design Space Definition

Referring to the title of the presented work, the design space parameters were limited merely to geometrical modification of the vehicle shape. Enabling inclusion of other types of parameters e.g. boundary condition settings were demarcated from the tool, although it would require little to add such functionality.

At VCC, CAD models are prior to CFD aerodynamics simulations treated in ANSA. Likewise, design parameters are defined in the ANSA software using built-in morphing functionality; including both morphing box and direct fit morphing methodology. An aerodynamics shape study might demand development of morphing strategies treating global changes to geometry and/or very detailed modifications. Parameterization can be extremely demanding depending on the complexity of the model and the desired result. It is important to define parameters so that high geometry quality is maintained and no discontinuities are observed throughout the entire design space.

Design of Experiments Method

Latin hypercube sampling (LHS) was chosen as the design of experiments method. With n design parameters there is an n -dimensional room of possible configurations. The number of simulated configurations should be minimized in order to avoid costly computations. DoE setups are statistical methods to reduce the number of samples through smart sampling. The accuracy of the solver approximation method *artificial neural networks* is increased for a fixed number of samples when generated using a DoE method rather than chosen randomly [9]. Alam et al. [13] investigated a number of different DoE methods used in conjunction with an artificial neural network. The study included a full factorial sampling, a random sampling, a central composite sampling and a LHS. The LHS method was found to outperform the other methods.

LHS is viable for any number of dimensions but for simplicity the theory of a two-dimensional LHS is described briefly here. The design space is divided into a number of rows and columns, whereafter data is randomly sampled exactly once from each row and each column. LHS has a smaller variance than a random sampling [14] and allows a reduced number of samples compared to the full-factorial design. A simple LHS with four samples can be seen in Figure 3. The figure visualizes the main concept of a LHS, where each row and each column has exactly one node.

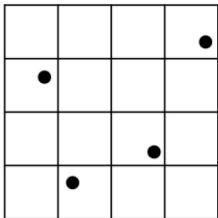


Figure 3. A latin hypercube sampling with four samples.

One of the drawbacks of LHS is that it does not reach minimal variance [14]. An improved and extended method is the maximin distance LHS which aims to find a LHS where the minimum distance between points is maximized, i.e. where the points are maximally spread out. This is done by adding a Monte-Carlo simulation to the LHS creation [13]. A large number of LHS designs are generated and the minimum distance between any two points is computed in each one. The design with the largest distance is chosen as the final design.

The default sample size N of the LHS design was chosen according to [11] as 2 points per dimension, i.e.

$$N = 2^m \quad (1)$$

where m is the number of parameters. Determining N through Equation 1 ensures an acceptable parameter space resolution with respect to neural networks.

Solver Approximation Method

An *artificial neural network* (ANN) was chosen as the solver approximation method. A solver approximation method is used to create an approximate relationship between the design parameters and

response behavior, in this case the behavior of aerodynamic drag and lift. The approximation can be modeled using a number of methods, out of which response surfaces and ANN:s are two of the more common ones. ANN:s are better at capturing non-linear effects as compared to response surfaces [11]. Due to the complex nature of the optimization problem at hand, non-linear effects are likely to occur. Therefore, an ANN was chosen as the solver approximation method.

An ANN is a computational model that mimics the behavior of an organic nervous system made up of neurons. The neurons of the system are replaced with computational nodes. An ANN can, as previously mentioned, be used to approximate non-linear functions; a desired trait in aerodynamic response modelling. A common network is the multi-layer feed forward network. This network is generally made up of several layers of nodes; one input layer, one or more hidden layers and one output layer [15]. An example of a multi-layer feed forward network is depicted in Figure 4.

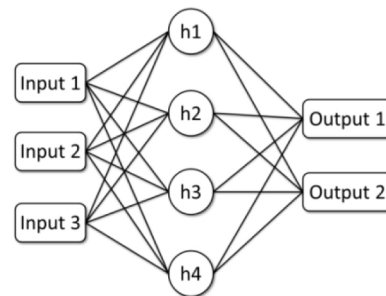


Figure 4. Schematic view of a single layer artificial neural network with three inputs, four hidden nodes and two outputs.

Signals are transferred from the input layer via the hidden nodes to the output layer. At each connection between nodes the signal is multiplied by a weight coefficient. The signals are summed up at the nodes and added to a bias [9]. The neural network is initialized by adjusting the weight and bias coefficients according to a database of inputs and corresponding outputs, this is known as training.

Training a neural network with an over-sized database can result in over-training. An over-trained network may closely reflect features of the training data such as noise and lose its generalization ability. Two common methods to avoid over-training are early stopping and regularization. Regularization allows the use of all samples for training, and for relatively small databases regularization yields better generalization than early stopping [16].

A single layer network with twice as many hidden nodes as parameters was chosen in accordance with results in [11]. The chosen neural networks training method was the commonly used Levenberg-Marquardt back-propagation method. For over-training control, the regularization method was selected.

Optimization Algorithm

Particle swarm optimization (PSO) was chosen as the optimization algorithm. There are many different methods of finding an optimal solution, ranging from traditional gradient-based algorithms to modern evolutionary algorithms. Gradient-based algorithms are in