

Adjoint Method for Aerodynamic Shape Improvement in Comparison with Surface Pressure Gradient Method

2011-01-0151

Published
04/12/2011Taeyoung Han
General Motors CompanyChris Hill and Shailesh Jindal
ANSYS, Inc.

Copyright © 2011 SAE International

doi:[10.4271/2011-01-0151](https://doi.org/10.4271/2011-01-0151)

ABSTRACT

Understanding the flow characteristics and, especially, how the aerodynamic forces are influenced by the changes in the vehicle body shape, are very important in order to improve vehicle aerodynamics. One specific goal of aerodynamic shape optimization is to predict the local shape sensitivities for aerodynamic forces. The availability of a reliable and efficient sensitivity analysis method will help to reduce the number of design iterations and the aerodynamic development costs. Among various shape optimization methods, the Adjoint Method has received much attention as an efficient sensitivity analysis method for aerodynamic shape optimization because it allows the computation of sensitivity information for a large number of shape parameters simultaneously. In the present study, the discrete adjoint equations to the ANSYS-Fluent pressure-based flow solver are formulated and solved in order to provide the aerodynamic shape sensitivities on the drag force for a simplified vehicle shape. The present study is intended to provide insights for aerodynamic shape improvement by observing the shape sensitivities and comparing the results for the diagnostic method based on the surface pressure gradients in the stream-wise and the transverse directions [1].

INTRODUCTION

Aerodynamic forces are the result of various complex viscous flow phenomena such as three-dimensional turbulent boundary layers on body surfaces, longitudinal vortices induced by three-dimensional boundary layer separation, and

high turbulence caused by flow separation. Understanding the flow phenomena and, especially, how the aerodynamic forces are influenced by the changes in vehicle body shape are very important for improving vehicle aerodynamics particularly for low drag shapes. Aerodynamic shape improvement currently requires repeated simulations of the flow around the vehicle by slightly perturbing the geometry of the 3D vehicle shape. The large number of flow simulations required can be significantly reduced if the adjoint method is applied to provide the direction for the shape change and the sensitivity of the shape change on the drag. A standard CFD code does not provide direct guidance on how changes to the vehicle shape will improve or worsen aerodynamic performance.

The main purpose of the present study is to demonstrate how the adjoint method can be used to provide the shape sensitivity and design direction to help improve a vehicle design. Simple test problems illustrate the method and the results are compared with the surface pressure gradient method that has been described previously in [1]. Much of the present paper is devoted to the adjoint method. An adjoint solver is a very attractive tool when applied to aerodynamic shape optimization of a complex 3-D vehicle shape. The traditional parametric optimization approach is not feasible due to the lack of a mathematical surface geometry to parameterize the complex 3D vehicle shapes. Even if it is feasible mathematically to define the 3D vehicle shape, the large number of shape parameters associated with the complex 3D vehicle shape will require a large number of flow simulations to obtain the design sensitivities of these shape parameters.

The idea of an adjoint method is centuries old, and interest in its use in the academic community for computational analysis has grown steadily in the past 30 years. Much of the development in this area in the US originated with Jameson [2], with the focus being on high-speed external aerodynamics of aircraft. The application to industrial engineering presents its own challenges and the technology is approaching the point where it can be applied to full-scale vehicle design. The challenge facing the designer is to determine how best to make adjustments to the vehicle shape to provide improved drag. At the same time these changes must not conflict with other design constraints such as roofline, trunk volume and styling. The adjoint method [3, 4, 5] allows the optimization of the vehicle shape under multiple constraints, such as the interior volume, comfort, styling, and various safety requirements.

The solution of an adjoint problem involves a computation that is on the scale of a conventional CFD solution for the problem. The adjoint method provides the design direction and the sensitivity information with only one solution of the adjoint equations, independent of the number of design variables. Most importantly the sensitivity of aerodynamic drag with respect to the shape of the vehicle can be extracted. An optimal change to the vehicle shape is achieved if it is made in proportion to the local shape sensitivity.

There are two main choices for implementing an adjoint solver, namely the continuous and the discrete approaches. The discrete adjoint was chosen here because it offers more consistent aerodynamic derivatives, especially in the presence of the use of wall functions. The discrete adjoint to the ANSYS Fluent pressure-based flow solver is formed. In order to demonstrate the concept of the adjoint method, a simple 2D airfoil problem [6] is described first and followed with a 3D vehicle shape. Simple morphing based on Bernstein polynomials defined relative to a set of control points is used. The sensitivity information for the mesh node locations is propagated to the control point locations. This has a smoothing effect on the sensitivity field, which can be noisy, and defines a smooth mesh deformation. A simplified vehicle shape, CAR3 [7] is used as an example geometry to test the effectiveness of the adjoint solution. The adjoint solution for drag sensitivity is solved and the shape sensitivity vector field examined. The results of this calculation are compared with the guidance provided with the surface pressure gradient method.

FORMULATION OF THE ADJOINT METHOD

A discrete adjoint solver is tied to the specific numerics that are used in the chosen conventional flow solver, although some simplifications can be made in the adjoint implementation. Prior to solving an adjoint problem the primary flow problem must be solved using standard

techniques. The adjoint problem is then solved relative to that particular flow solution.

A high-level view of the adjoint is available when the flow problem is considered at the system level. There are three ingredients:

1. The inputs to the system. This includes all boundary conditions used to define the problem, material properties of the fluid, modeling parameters such as those in a turbulence model, and the definition of the mesh. The mesh definition is of course a rather large data set for external aerodynamic modeling of vehicles. This set of variables is denoted here by a vector $\underline{C}$. Specifically excluded from this list are variables that control the solution-advancement process to the final flow solution.

2. The discretized nonlinear form of the Navier-Stokes equations and turbulence equations that are satisfied at

convergence. Let $\underline{q}^i$ denote the vector of flow variables in the i th cell of the computational grid. Within any cell the flow variables includes pressure and velocity, as well as for example k , and ε for a turbulence model. At convergence a discrete set of governing equations is satisfied that is a reflection of the flow physics being modeled. While there is much richness in how this is done it can be represented symbolically on the μ -th cell as $\underline{R}^\mu(\underline{q}^0, \underline{q}^1, \dots, \underline{q}^{N-1}; \underline{C}) = 0$ where there are N cells, and the number of conditions on each cell equals the number of flow variables. The conditions on any cell will usually involve the flow variables at 1st and 2nd neighbors, and possibly more.

3. An engineering observation of interest such as lift or drag. Once a conventional flow solution has been determined, engineering quantities of interest can be computed by reducing the flow results to a single scalar

value, such as drag. Here $J(\underline{q}^0, \underline{q}^1, \dots, \underline{q}^{N-1}; \underline{C})$ denotes such an observation, noting its dependency on both the flow state and the inputs specified for the problem.

Given such a system the goal is to determine the derivative of the engineering observation of interest with respect to any and all inputs to the problem, with the shape derivatives being of particular interest and value. This goal is in contrast to the *tangent problem*, in which the derivative of the flow solution is computed with respect to a single input.

It is first noted that if an adjustment δc_j is made to the inputs, then there will be an adjustment in the flow variables that we denote here by δq_j^v . The adjustment to the objective function, J , is then

$$\delta J = \frac{\partial J}{\partial q_j^v} \delta q_j^v + \frac{\partial J}{\partial c_j} \Big|_{q_j^v} \delta c_j \quad (1)$$

where a summation over the indices j and v is implied, and $\Big|_{q_j^v}$ denotes that the derivative is computed with the flow state held constant.

The difficulty with this form is that the variation in the flow field depends upon the adjustment that is made to the inputs. The relationship between these two is given by the linearization of the Navier-Stokes equations:

$$\frac{\partial R_i^\mu}{\partial q_j^v} \delta q_j^v = - \frac{\partial R_i^\mu}{\partial c_j} \Big|_{q_j^v} \delta c_j \quad (2)$$

At first glance it would appear that in order to compute the change in the observation, it is necessary to pick a change to the controls, δc_j , form the right hand side of (2), solve the (large) linear system for the modification to the flow state, δq_j^v , and thereby construct δJ using (1). This is the so-called tangent problem and can be a useful strategy for some classes of analysis. A different approach is adopted, namely the adjoint method, to meet the present needs.

The crux of the adjoint method is the introduction of an auxiliary field, namely the adjoint field $\bar{q}_i^\mu$. Note the over bar here that is used to distinguish this field from the flow field. Also note the one-to-one correspondence between the number of adjoint variables and the number of equations that are solved. The field is chosen to satisfy the equation

$$\frac{\partial R_i^\mu}{\partial q_j^v} \bar{q}_i^\mu = \frac{\partial J}{\partial q_j^v} \quad (3)$$

Much of the computational effort in an adjoint solver is in the construction and solution of this system.

It is of note that the matrix on the left hand side is the *transpose* of the system Jacobian. The particular system, (3), is not derived from anywhere but is actually a definition for the adjoint field. This choice is made in anticipation of a chain of substitutions:

$$\begin{aligned} \delta J &= \frac{\partial J}{\partial q_j^v} \delta q_j^v + \frac{\partial J}{\partial c_j} \Big|_{q_i^\mu} \delta c_j \\ &= \bar{q}_i^\mu \frac{\partial R_i^\mu}{\partial q_j^v} \delta q_j^v + \frac{\partial J}{\partial c_j} \Big|_{q_i^\mu} \delta c_j \\ &= \left\{ \frac{\partial J}{\partial c_j} \Big|_{q_i^\mu} - \bar{q}_i^\mu \frac{\partial R_i^\mu}{\partial c_j} \Big|_{q_i^\mu} \right\} \delta c_j \end{aligned} \quad (4)$$

The result is an explicit expression for the derivative of the objective function with respect to the control variables, with the variation in the flow itself eliminated. The price for obtaining this information is that a specific adjoint problem must be solved. The benefit is that with a one-time solution of the adjoint problem, the derivative of the objective function with respect to any and all input variables is made available. This includes the sensitivity and the design direction with respect to the locations of the grid nodes that define the geometry of the problem.

Focusing solely on geometry variations, we denote the subset of the variables $\mathcal{C}$ that define node locations by x_i^n , where n is the index for the boundary node and the subscript i denotes the coordinate direction. From (4) the change in the objective function can be correlated simply with mesh node position variations as

$$\delta J = \sum_n \bar{w}_i^n \delta x_i^n \quad (5)$$

where

$$\bar{w}_i^n = \left\{ \frac{\partial J}{\partial x_i^n} \Big|_{q_i^\mu} - \bar{q}_j^\mu \frac{\partial R_j^\mu}{\partial x_i^n} \Big|_{q_i^\mu} \right\} \quad (6)$$

The coefficients $\bar{w}_i^n$, once computed, provide the sensitivity of the drag on a vehicle to shape changes. These coefficients define a design direction vector field on the vehicle surface. In locations where the vector field is large the vehicle drag is influenced strongly by the shape. Small changes of the vehicle surface in that location will have a relatively large effect on drag. In contrast, at locations where the magnitude of the vector is small the drag is not influenced strongly by making changes to the local vehicle shape.

It is possible to interpret the adjoint solution fields themselves as providing a map of how receptive the complex aerodynamic system is to external influences of many different kinds [8]. The main interpretation for the present