

Development of CFD Shape Optimization Technology using the Adjoint Method and its Application to Engine Intake Port Design

Shigefumi Tokuda, Masato Kubota and Yasushi Noguchi
Toyota Motor Corp

ABSTRACT

Computational fluid dynamics (CFD) shape optimization technology is playing an increasingly significant role in the development of products that satisfy various demands, including trade-off relationships. It offers the possibility of designing or improving product shape with respect to a given cost function, subject to geometrical constraints. However, conventional CFD shape optimization technology that uses parametric shape modification has two following issues: (1) expensive computational cost to obtain the final shape, (2) performance variations of the obtained shape depends on the skill or experience of the designer who determined the locations to be modified.

In this study, to resolve those problems, an efficient shape optimization technology was developed that uses the adjoint method to perform sensitivity analysis of a cost function on the design parameters. It is composed of a combination of topology optimization and surface geometry optimization. Firstly, topology optimization is used to roughly derive a base shape in the constrained design space. Next, surface geometry optimization is used to further modify the base shape according to the sensitivity distributions of the cost function to each vertex on the surface geometry. This technology was then applied to the design of an engine intake port. The cost function was defined as the balance between maximizing the intake flow rate and in-cylinder tumble moment. As a result, topologically optimized shapes with various performance aspects according to the weighting factor of the cost function were obtained. Then, the topologically optimized shapes were fine-tuned by surface geometry optimization.

CITATION: Tokuda, S., Kubota, M. and Noguchi, Y., "Development of CFD Shape Optimization Technology using the Adjoint Method and its Application to Engine Intake Port Design," *SAE Int. J. Engines* 6(2):2013, doi:10.4271/2013-01-0969.

INTRODUCTION

As issues related to the environment and the energy continue to gain prominence, automakers are facing demands for high quality and safety, reduced emissions of CO₂ and the like, improved performance, and lower product cost. To achieve these goals within a limited development lead-time, it is important to design good-quality shapes in a short period of time at the initial design stage before prototyping to prevent the need for re-design after evaluation. The CFD shape optimization technology is considered to be an effective means of deriving shapes in advance to meet this aim and to satisfy the large number of target performance requirements, including those that have a trade-off relationship.

As shown in the flowchart in [Figure 1](#), the conventional CFD optimization technology requires the designer to specify design parameters (i.e., locations where the shape is changed) and to carry out iterations of parametric changes to the shape. As an example, [Figure 2](#) shows the results of shape

optimization for an engine intake port. In this example, the cost functions are the achievement of a tumble ratio of 1 or more while minimizing pressure loss across the intake port. Each diamond symbol (◇) in the figure indicates the results of CFD analysis for a modified shape. The figure demonstrates that the conventional shape optimization requires a large number of numerical iterations until derivation of the final shape. Furthermore, since the designer pre-determines the areas of shape modification, the ultimate performance of the part varies widely depending on the experience of the designer. These two issues obstruct the implementation of optimization method to a major application in the design development process of products.

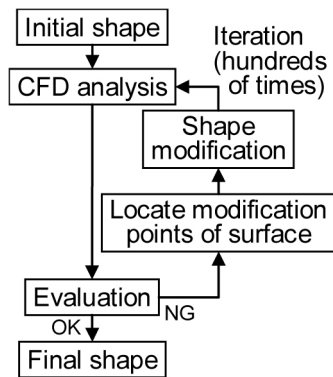


Figure 1. Process of conventional CFD shape optimization

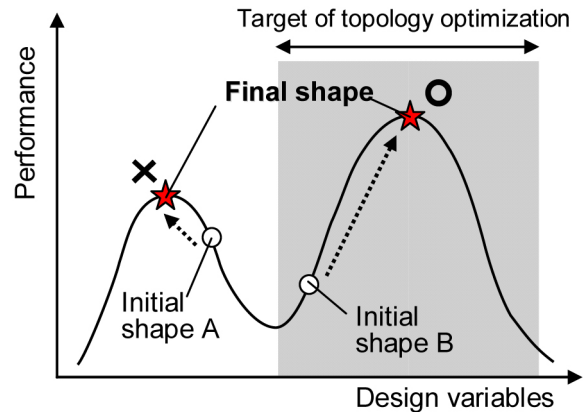


Figure 3. Schematic image of initial shape dependency

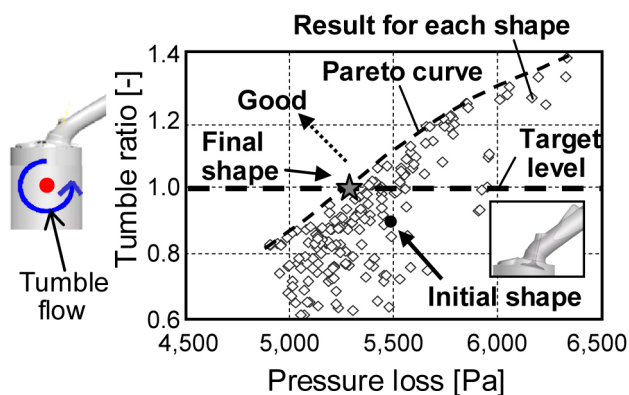


Figure 2. Results of conventional CFD shape optimization

To address these issues, surface geometry optimization technology that uses the adjoint method has been researched [1, 2, 3, 4]. This method is capable of theoretically calculating the degree of performance change (i.e., the shape change sensitivity) due to modifications made to the object shape, before the shape is actually changed. However, surface geometry optimization technology using the adjoint method changes shapes based on the gradient of target cost functions with respect to the shape change from a base shape. Therefore, as shown in Figure 3, selection of a certain starting base shape may only result in the localized optimum solution, preventing the derivation of a global optimum shape that provides superior performance.

In the field of structural analysis, there has been a research focusing on the technology capable of optimizing the topology. The topology optimization allows modifications to different shapes such as adding holes and other geometric features and it has been applied to various design optimization problem with high degree of freedom [5, 6, 7, 8].

In the field of fluid analysis, an example of its application to Stokes flow has been reported. This proposes a method to consider an object placed in a fluid as a porous body and to express its shape with porosity [9]. Other reports have described its application to design problems related to flow path surfaces and the optimization of object shapes in a fluid, focusing on viscous flows with a comparatively small Reynolds number (Navier-Stokes equations) [10, 11, 12, 13, 14].

More recent reports have proposed methods of shape optimization that use the adjoint method to calculate the changes in porosity sensitivity [15, 16]. Such topology optimization technology using the adjoint method is capable of deriving general shapes within the design space defined by spatial constraints, and it does not depend on experience or preconceived ideas of the designer. This technology has the potential to derive initial shapes (such as the shape shown in the right peak in Figure 3) to which surface geometry optimization can be applied to achieve the final shape.

In this way, surface geometry optimization technology or topology optimization technology using the adjoint method allows cost functions to be set with a high degree of freedom, and has general applicability to the design of various parts. However, the examples of research described above had to incorporate adjoint method programs into open source CFD software. This is because the implementation of the adjoint method and the setting of cost functions require modifications to the internal structure of the solver. However, since the programs of general purpose commercial CFD software are not openly available, it is difficult to incorporate the adjoint method into the software. In addition, since design uses limited cost functions such as pressure loss, drag force, and

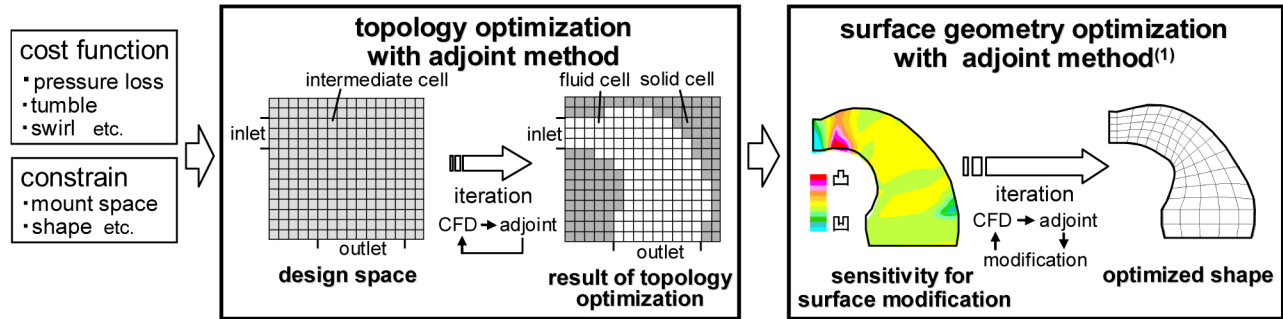


Figure 4. Process of CFD shape optimization by combination of topology and surface geometry optimization with adjoint method

lift force, the adjoint method has yet to find large-scale applications in design development.

Therefore, the research described in this paper utilized the merits of the adjoint method (i.e., the freedom to set cost functions) to develop optimization technology capable of performing shape optimization in combination with any CFD software. The aim of the research was to enable the application of this method to the design development of various parts. Figure 4 shows the process flow of the developed optimization technology that combines both topology and surface geometry optimization technology using the adjoint method.

COMPUTATIONAL METHOD

Design Parameters in Optimization using the Adjoint Method

The CFD shape optimization using the adjoint method calculates the change of the cost functions (i.e., the performance aspects to be enhanced) due to the design parameters modifications. Here, the design parameter in topology and surface geometry optimization using the adjoint method and those methodologies are described.

As shown in Figure 4, the first step of topology optimization using the adjoint method fills the design space defined by the spatial constraints with cells that initially do not distinguish between fluid or solid material (i.e., the cells are all given a certain porosity). After CFD analysis, adjoint analysis is performed to determine the sensitivity of cost function to the porosity of each cell. In other words, the porosity of each cell is the design parameter, and this step is determining the distribution of fluid cells (low porosity) and solid cells (high porosity), that enhances the cost function. The porosity of each cell is then changed according to this distribution. This procedure is repeated to divide the design space into fluid and solid areas. The topologically optimized shape is obtained once the porosity stops changing.

In contrast, in surface geometry optimization, the sensitivity distribution is determined based on the changes to the surface geometry, while enhancing the cost function.

Design parameters here are the coordinates of the vertices that make up the surface geometry.

Sensitivity Analysis using the Adjoint Method

The following section describes the method for calculating the sensitivity of the cost function to design parameters. The continuity equation (1) and the Navier-Stokes equation (2) below are the governing equations for an incompressible steady-state flow field, including a porous body.

$$\frac{\partial \rho u_i}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial \rho u_i u_j}{\partial x_j} + \frac{\partial p}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} - \rho \alpha u_i = 0 \quad (2)$$

$$\tau_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu \delta_{ij} \frac{\partial u_k}{\partial x_k} \quad (3)$$

Here, ρ is the density of the fluid, u is the velocity, p is the pressure, τ is the viscous stress tensor, α is the porosity of the fluid, μ is the dynamic coefficient of viscosity, and δ_{ij} is the Kronecker delta.

Here, if the design parameters that are changed during the shape optimization (i.e., the coordinates x_n and the cell porosity α_n) are denoted as D_n , and the physical quantities (i.e., the velocity $u(x_n, \alpha_n)$, the pressure $p(x_n, \alpha_n)$, and other quantities) are denoted as $\mathbf{U}(D_n)$ and are used to rationalize the continuity and Navier-Stokes equations as $\mathbf{R}(\mathbf{U}(D_n))$, $D_n=0$ ($n=1, 2, \dots$ total number of design parameters), then the variation of the governing equation $\mathbf{R}=0$ with respect to the changes in the design parameters D_n can be expressed as