Commun. Comput. Phys.
doi: 10.4208/cicp.OA-2023-0020

Vol. **33**, No. 5, pp. 1381-1408
May 2023

Solution Remapping Method with Lower Bound Preservation for Navier-Stokes Equations in Aerodynamic Shape Optimization

Bin Zhang[1], Weixiong Yuan[1], Kun Wang[1], Jufang Wang[2] and Tiegang Liu[1,*]

[1] *LMIB and School of Mathematical Sciences, Beihang University, Beijing 100191, China.*

[2] *Computational Aerodynamic Research Institute, China Aerodynamics Research and Development Center, Mianyang 621000, China.*

Received 15 January 2023; Accepted (in revised version) 30 March 2023

Abstract. It is found that the solution remapping technique proposed in [Numer. Math. Theor. Meth. Appl., 2020, 13(4)] and [J. Sci. Comput., 2021, 87(3): 1-26] does not work out for the Navier-Stokes equations with a high Reynolds number. The shape deformations usually reach several boundary layer mesh sizes for viscous flow, which far exceed one-layer mesh that the original method can tolerate. The direct application to Navier-Stokes equations can result in the unphysical pressures in remapped solutions, even though the conservative variables are within the reasonable range. In this work, a new solution remapping technique with lower bound preservation is proposed to construct initial values for the new shapes, and the global minimum density and pressure of the current shape which serve as lower bounds of the corresponding variables are used to constrain the remapped solutions. The solution distribution provided by the present method is proven to be acceptable as an initial value for the new shape. Several numerical experiments show that the present technique can substantially accelerate the flow convergence for large deformation problems with 70%-80% CPU time reduction in the viscous airfoil drag minimization.

AMS subject classifications: 49Q10, 65M12, 65M22

Key words: Aerodynamic shape optimization, solution remapping technique, direct discontinuous Galerkin method, lower bound preservation, Navier-Stokes equations.

---

*Corresponding author. *Email addresses:* liutg@buaa.edu.cn (T. Liu), binzhang@buaa.edu.cn (B. Zhang), yuanweixiong@buaa.edu.cn (W. Yuan), wangkun@buaa.edu.cn (K. Wang), wangjufang@buaa.edu.cn (J. Wang)

1 Introduction

In recent decades, aerodynamic shape optimization (ASO) design methods based on computational fluid dynamics (CFD) have been developed rapidly. A large number of intermediate shapes in the optimization process need to be simulated to the steady state to evaluate their aerodynamic performance. As a result, the 2-nd order finite volume method with a cheap computational cost is widely used to solve the flow field governing equations to reduce the time required for each intermediate shape in industrial applications [4, 9, 11]. Recently, high-order numerical methods represented by the (direct) discontinuous Galerkin [5,6] methods are widely applied in flow field simulation to meet the requirements of higher precisions [2,10,21], but their expensive computational cost [22,23] prevents them further development from industrial applications.

The selection of the initial value of flow field for intermediate shapes would significantly affect the efficiency of optimization. There are two popular methods of the initial value construction. One is to use the freestream flow, that strategy always converges to the steady-state solution, but usually with extremely poor efficiency. The other is to copy the converged solution of the previous shapes directly [14,27,30], such an approach may occasionally result in lower efficiency, we think possibly because the shape deformation is not taken into consideration. We know that the evolution of the adjacent intermediate shapes in the optimization process is usually small, this implies that the steady-state flow field of the next intermediate shape is a small deviation from that of the present shape. As a result, it is possible to accelerate the flow convergence by wisely employing the steady-state solution of the previous shape to construct the initial values of the new shapes [3].

Recently, the solution remapping technique based on the above concept has been successfully applied to the ASO with great acceleration performance [19, 20] and can save 70%-80% CPU time for inviscid flow. This technique was first proposed based on the finite volume method for solving the Euler equations by Wang et al. [20] with the requirement of shape deformation limited in about one-layer mesh size. Later, Wang and Liu extended this technique to high order DG methods [19] with a maximum-and-minimum-preserving limiter applied to modify the remapped solutions in order to relax its limitation of less one-layer mesh size on the shape deformation. We find that negative pressure in the remapped solutions can occur when directly applied to Navier-Stokes equations, even though the conservative variables are within the reasonable range for the large shape deformations.

In this work, we shall develop a new solution remapping technique based on the above-mentioned idea to the Navier-Stokes equations for viscous flow. Since there are extremely thin boundary layer meshes near the airfoil in viscous flow, the shape deformations usually exceed several boundary layer meshes, which means that the shape deformations are always relatively large deformations compared to the boundary layer meshes. Therefore, in order to make this technique available to the N-S equations, we have to break through the boundary layer mesh size limitation for large deformation