An improved viscous / inviscid velocity decomposition method

Deborah O. Edmund*, Kevin J. Maki, and Robert F. Beck
University of Michigan, Ann Arbor, Michigan 48109, USA
[doedmund, kjmaki, rbeck] @umich.edu
***Presenting Author**

Introduction

Modeling hydrodynamic forces on submerged or floating bodies is integral to the design process. Many hydrodynamic solution methods exist, ranging from the geometrically simplified strip theory, to inviscid approaches and the fully nonlinear unsteady Reynolds-Averaged Navier-Stokes (RANS) solvers. The former approaches are generally less expensive to use, but neglect various aspects of the relevant physics including viscous effects, and often wave breaking. RANS solvers include viscosity and can handle wave breaking; however they are generally too expensive to be widely utilized at the design stage. The current research is focused on finding a strategic combination of the RANS and potential flow solvers to deliver the benefits of each in a unified methodology. The proposed solver is based on the work of Kim et al. (2005), which utilizes a Helmholtz type velocity decomposition to describe the total velocity field $\overrightarrow{u}$ as the sum of an irrotational component $\nabla\Phi$ and a complementary component $\overrightarrow{w}$:

$$\overrightarrow{u} = \nabla\Phi + \overrightarrow{w} \tag{1}$$

The potential velocity $\nabla\Phi$ is determined in the infinite fluid domain using a standard boundary element method. The decomposed velocity is inserted into the Navier-Stokes equations to derive a modified form of the Navier-Stokes equations, referred to as the complementary RANS equations. The complementary RANS equations govern the complementary velocity field, and are solved using a finite volume method. The new thrust of this research is to make the complementary RANS solver achieve computational savings by improving the potential flow solution so the complementary velocity becomes negligible just outside the boundary layer, and the computational domain can therefore be reduced. Lighthill's equivalent source method distributes sources on the body surface to push the inviscid streamlines outward to match the viscous boundary layer thickness (1958). This equivalent source method has since been called the transpiration velocity to reflect the apparent flow through the body surface and is used here to improve the potential velocity field.

The ultimate applications of the proposed method include three-dimensional bodies moving at high Reynolds numbers through a free surface and bodies at or near a free surface with waves. The current work has been initiated by developing and demonstrating the proposed method on two-dimensional fully submerged bodies at laminar-flow Reynolds numbers.

Methodology

The governing equations for the complementary component $\overrightarrow{w}$ are found by substituting the decomposed velocity into the mass and momentum balances for steady flow with constant density and viscosity, leading to the continuity equation and steady complementary Navier-Stokes equations as follows:

$$\nabla\cdot\overrightarrow{u} = \nabla\cdot\nabla\Phi + \nabla\cdot\overrightarrow{w} = 0 \tag{2}$$

$$\nabla\cdot\overrightarrow{u}\,\overrightarrow{w} + \nabla\cdot\overrightarrow{u}\nabla\Phi = -\frac{\nabla p}{\rho} + \nu\nabla^2\overrightarrow{w} \tag{3}$$

where p is pressure, ρ is density, and ν is kinematic viscosity. The current work is focused on steady flows, so the time derivative in the Navier-Stokes equations is neglected. To satisfy the no-slip boundary condition, the total velocity must equal zero on the fixed body boundary, indicating that $\overrightarrow{w} = -\nabla\Phi$ on the body boundary. The total velocity is assumed to equal the irrotational or potential velocity far from the body and outside the rotational wake, so $\overrightarrow{w} = 0$ is enforced on the inlet and farfield boundaries. The complementary velocity outlet boundary condition and the pressure boundary conditions on the body, inlet, and farfield are taken to be zero gradient. The pressure is set to zero at the outlet to serve as the reference pressure.

The potential velocity field is determined using a two-dimensional constant strength source panel method. The source strengths are determined by satisfying the Neumann body boundary condition of non-penetration or by setting the normal velocity on each panel equal to the transpiration velocity in the improved solver. Lighthill defines the two-dimensional transpiration velocity $v_{trans} = d(U_e \delta^*)/dx$, where U_e is the external velocity and δ^* is the displacement thickness (Lighthill 1958). The second boundary condition imposed on the potential velocity is the radiation condition, which ensures that the undisturbed velocity $\overrightarrow{U}_\infty$ is recovered far from the body - the panel source solution satisfies this naturally.

The potential flow equations and complementary RANS equations are implemented as a complementary RANS solver in the computational fluid dynamics environment Open Source Field Operation and Manipulation (OpenFOAM). The steady incompressible viscous flow solver simpleFoam was chosen as the base RANS solver since it represents an industry standard steady RANS solver. The simpleFoam solver uses the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm as described by Patankar (1980), and will be referred to as the RANS solver.

The complementary RANS equations are discretized using the finite-volume method in the same manner as the conventional RANS equations. The only difference is the addition of a source term due to the convection of the potential velocity by the total velocity. Both the RANS solver and the complementary RANS solver use second order Gaussian integration with linear interpolation of the cell center values to the face centers for the gradient and Laplacian terms. The Laplacian terms also use an explicit non-orthogonal correction for the surface normal gradient. The divergence terms are discretized by Gaussian integration, but use a bounded first/second order linear upwind interpolation scheme that uses Gaussian integration and linear interpolation for its gradient scheme. Overall, the equation discretization is formally of second order.

Results

Laminar flow over a NACA 0010 airfoil at a Reynolds number (Re) of 7900, based on the chord (L) of the airfoil, has been studied to verify the present implementation of the complementary RANS solver. The computational domain for the NACA 0010 airfoil has an entrance length of two and a half chord lengths, width of five chord lengths with the airfoil at the center, and an exit length of five chord lengths. There are 204 panels on the airfoil. The grid uses a c-topology with 184×164 nodes, which are concentrated around the airfoil, wake, leading edge, and trailing edge using geometric sequence expansions. All of the cases presented in this work were run until the residuals, defined as a scaled L1 norm, were reduced to below 1×10^{-10} for pressure and 1×10^{-12} for velocity.

The velocity profiles from the solvers of Kim et al. (2005), the RANS solver, and complementary RANS solver at $x/L = 0.54$ and $x/L = 1.0$ are shown in Figure 1. At $x/L = 0.54$, the four velocity profiles are nearly indistinguishable with an RMS error of 0.29% between the RANS and complementary RANS solvers. The velocity profiles at the trailing edge of the airfoil ($x/L = 1.0$) match well, but reveal minor discrepancies between the solvers from Kim et al. and the current solvers. It is evident that the difference between the current RANS and complementary RANS solvers is smaller than the difference between those of Kim et al. The RMS error at the trailing edge between the currently proposed solvers is 0.36%. Both of the complementary RANS solvers exhibit a small oscillation in the trailing edge velocity profile just above the centerline. Kim et al. was able to reduce this oscillation to the relatively small size shown by adding a source in the trailing edge of the airfoil. No modifications have been made to the potential solution to reduce this oscillation in the current solver.

The NACA 0010 airfoil results show that the complementary RANS solver essentially recreates the RANS solver solution and matches the results of Kim et al. quite well, but the computational expense has been increased. Kim et al. also observed this increase in computational expense, which is due to the calculation of the potential velocity field and the additional term in the complementary RANS solver.

Figure 2 shows velocity profiles at $x/L = 0.54$ and $x/L = 1.0$ for the RANS solver on the full domain denoted as $2.5L$ and a reduced domain that extends only 0.6 chord lengths in the upstream and farfield directions denoted as $0.6L$. In addition, the solution for the complementary RANS solver and preliminary results for the complementary RANS solver with transpiration velocity are provided on the reduced domain. The transpiration velocity is determined by the Blasius flat plate solution for these preliminary results. Both solvers exhibit a significant variation from the full domain results on the reduced domain without the use of transpiration velocity. The reduced domain complementary RANS results differ from the full domain RANS results less severely than the reduced domain RANS results due to the velocity boundary conditions on the inlet and farfield boundaries which set the total velocity equal to the potential velocity rather than the free-stream velocity as it is in the RANS solver. Including the transpiration velocity significantly reduces the error caused in the complementary RANS solver by reducing the domain size, and captures the velocity profiles quite well.

Conclusions and Future Work

The complementary RANS solver presented here compares well with an industry standard RANS solver, and the results of Kim et al. (2005), but fails to realize computational savings. Without using a transpiration velocity to improve the potential velocity field, reducing the domain to achieve computational savings decreases the accuracy of the results. Including the approximate form of a transpiration velocity based on a flat plate formulation in the NACA 0010 airfoil solution significantly increased the accuracy on the reduced domain. Given the promising preliminary results of the transpiration-velocity method, current work is focused on determining the transpiration velocity from the velocity field. The complementary solver will be extended to handle turbulent flows and free-surface problems will be studied.

Acknowledgements

This research is supported by the Department of Defense (DOD) through the National Defense Science & Engineering Graduate Fellowship (NDSEG) program.

References

Kim, K., Sirviente, A. I., and Beck, R. F., "The complementary RANS equations for the simulation of viscous flows," *International Journal for Numerical Methods in Fluids*, Vol. 48:199-229, 2005.

Lighthill, M. J., "On displacement thickness," *Journal of Fluid Mechanics*, Vol. 4:383-392, 1958.

Patankar, S. V., *Numerical Heat Transfer and Fluid Flow, Series in Computational and Physical Processes in Mechanics and Thermal Sciences*, Taylor & Francis, p. 126-129, 1980.

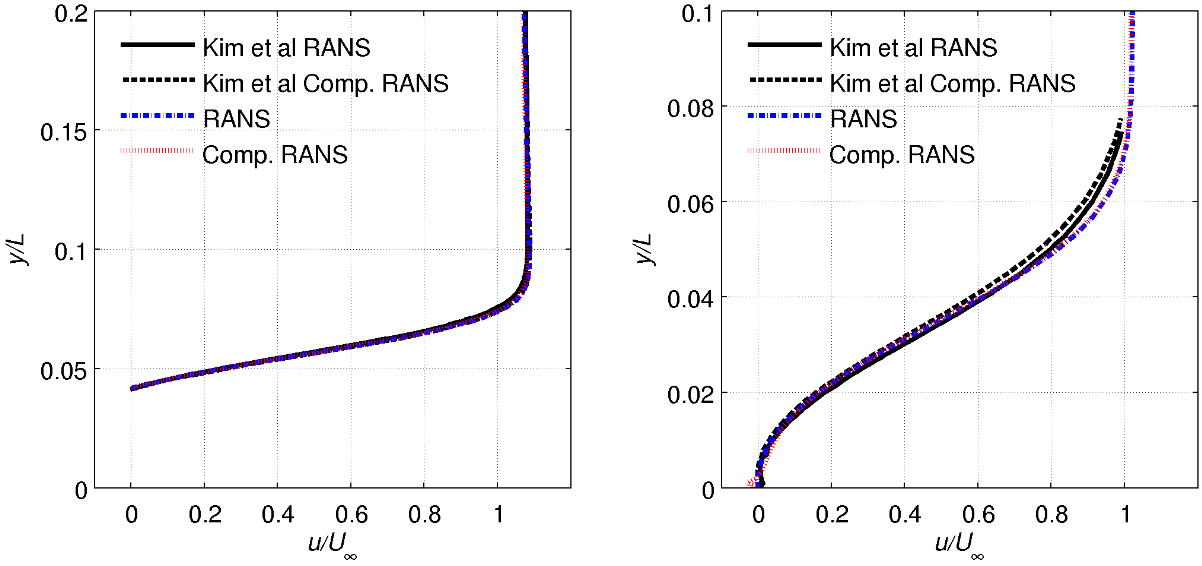

Figure 1: Velocity profiles from the RANS and complementary (Comp.) RANS solvers from Kim et al. and the current work for flow around a NACA 0010 airfoil with Re = 7900 at (Right) $x/L = 0.54$ and (Left) $x/L = 1.0$ (Kim et al. 2005).

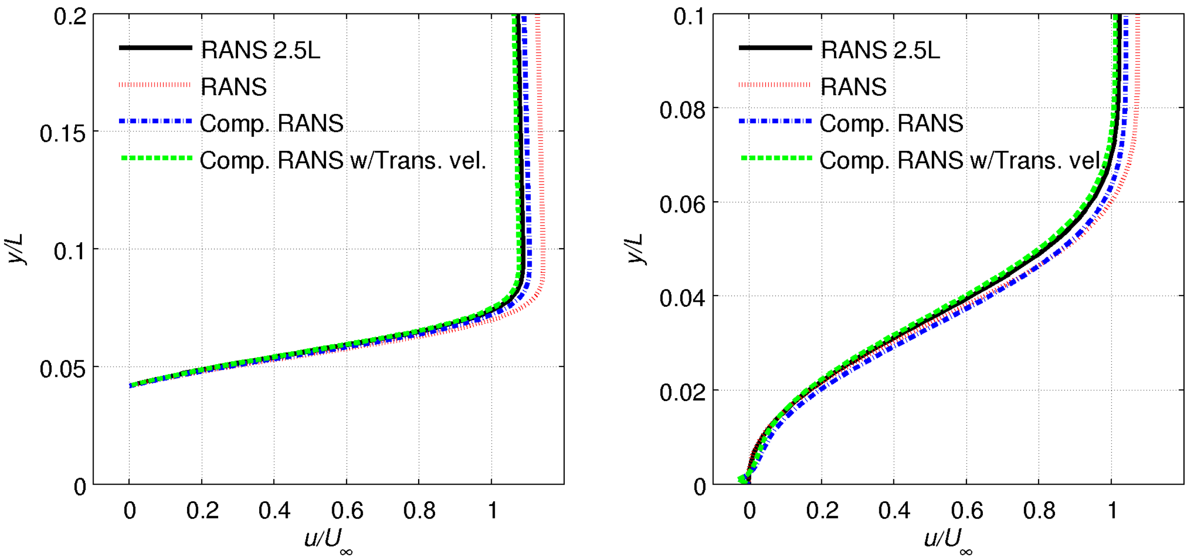

Figure 2: Velocity profiles from the RANS solver on the full domain ($2.5L$) and the RANS solver, complementary (Comp.) RANS solver, and complementary (Comp.) RANS solver with transpiration velocity on the reduced domain ($0.6L$) for flow around a NACA 0010 airfoil with Re = 7900 at (Right) $x/L = 0.54$ and (Left) $x/L = 1.0$.