Numerical simulation of airfoil flow based on the Navier-Stokes equations

D. I. Kushnir[1]

[1]Institute of Hydromechanics of the National Academy of Sciences of Ukraine, Kyiv, Ukraine

kushnirdima2000@gmail.com

The study of flow around an airfoil remains highly relevant for problems in aerohydrodynamics. Accurate reproduction of aerodynamic processes is critically important for predicting lift, drag, and motion stability, especially under conditions of complex unsteady flow, external disturbances, structural elasticity, and the presence of a ground-effect surface [1]. Computational Fluid Dynamics (CFD) methods significantly expand the possibilities for investigating flow phenomena; however, they require a careful approach to problem formulation, computational mesh generation, specification of boundary conditions, selection of numerical solution algorithms, and subsequent validation of the obtained results. The purpose of this study is to consistently perform the complete CFD modeling cycle for airfoil flow simulation, including mathematical formulation of the problem, selection and implementation of the numerical method, computational mesh generation, specification of boundary conditions, and configuration of numerical solution algorithms.

For the mathematical description of the airfoil flow, the unsteady Navier-Stokes equations for a viscous incompressible fluid are employed. At the boundaries of the computational domain, boundary conditions corresponding to an undisturbed incoming flow are imposed, while no-slip conditions are applied on the body surface. The resulting system of algebraic equations is solved using iterative methods with the application of a geometric-algebraic multigrid pressure solver (GAMG) and an iterative velocity solver, while the coupling between pressure and velocity fields is ensured by the PIMPLE algorithm (OpenFOAM CFD package).

In constructing the numerical model of the airfoil flow, a two-dimensional (2D) formulation is adopted in order to reduce computational complexity. This approach is physically justified for moderate Reynolds numbers (on the order of 10,000 to 100,000), where the flow is predominantly characterized by large-scale quasi-two-dimensional structures and three-dimensional effects are not dominant [2, 3].

The simulation is performed using Direct Numerical Simulation (DNS), which allows direct resolution of the flow structure without the use of turbulence models. In comparison with approximate approaches such as LES (Large Eddy Simulation) and URANS (Unsteady Reynolds-Averaged Navier-Stokes), DNS provides the highest level of physical completeness of the solution, although it imposes significantly higher requirements on computational mesh resolution. In the present work, a simplified (non-ideal) mesh is employed; however, for the considered Reynolds number range this does not lead to significant accuracy loss, since the flow is dominated by large-scale vortex structures that are reproduced correctly even at limited spatial resolution.

The structured computational mesh is generated using the finite volume method, which provides integral discretization of the Navier-Stokes equations over control volumes. Such an approach leads to the formation of a discrete algebraic system of equations in which continuous differential operators are replaced by corresponding numerical approximations, while the mesh geometry directly determines the structure and properties of the resulting system. As part of the numerical model verification, a mesh convergence study was conducted in order to determine the level of spatial discretization at which further mesh refinement no longer produces significant changes in the integral solution characteristics. It was shown that insufficient mesh refinement results in sensitivity of aerodynamic coefficients to mesh resolution, whereas after reaching a certain discretization level, changes in the results become limited and remain within the prescribed error tolerance.

The obtained aerodynamic coefficients (lift, drag, and moment coefficients) were validated through comparison with experimental data [4]. Good agreement of the results (less than 10 percent) was achieved within the range of moderate angles of attack (up to 7.5 degrees), confirming the correctness of the mathematical formulation and the applied numerical method. Analysis of the numerical solution structure of the Navier-Stokes equations demonstrates a significant dependence of the flow behavior on the Reynolds number and angle of attack. Since the problem is unsteady, aerodynamic characteristics are considered as time-dependent functions. The obtained time series demonstrate a transition toward a quasi-steady regime with oscillations around mean values, reflecting the dynamic properties of the solution of the discretized system of equations.

[1] Panchenkov A. N., Hydrodynamics of a hydrofoil, Naukova Dumka, Kyiv, 1965.

[2] Hemanth A., Kushal K. S., Varun S. S., Aerodynamic Study on Low Reynolds Number Aerofoil, *ACS Journal for Science and Engineering*, **4** (2024), no. 1, 39–57.

[3] Ahmed M. R., Narayan S., Zullah M. A., Lee Y. H., Experimental and Numerical Studies on a Low Reynolds Number Airfoil for Wind Turbine Blades, *Journal of Fluid Science and Technology*, **6** (2011), no. 3, 357–371.

[4] Ohtake T., Nakae Y., Motohashi T., Nonlinearity of the Aerodynamic Characteristics of NACA0012 Aerofoil at Low Reynolds Numbers, *Journal of the Japan Society for Aeronautical and Space Sciences*, **55** (2007), no. 644, 439–445.