Numerical Investigation to Optimize Geometry of Hydrofoil Shaped Container Drone

Anashwara Binod, Bobin Saji George, Deepak G. Dilip, Ebel Philip Varghese, R. Abhishek, and Arjunlal Jawaharlal

1 Introduction

A lifting surface, or foil, that acts in water is known as a hydrofoil. They are similar in look and purpose to airfoils used by airplanes. Boats that utilize hydrofoil technology are also simply named hydrofoils. As a hydrofoil craft develops speed, the hydrofoils lift the boat's hull out of the water, minimizing drag and permitting greater speeds. The lifting body that produces high lift and low drag is a hydrofoil. Because it can now produce quicker and more accurate results for various flow parameters around the geometry or model, computational fluid dynamics (CFD) has recently grown to be very important to researchers. It is simpler to determine the flow on the hydrofoil model in detail using CFD calculations. Also, it is cost effective and it gives gaining the work time instead of the experimental research and have significant prices. Under addition, variation in flow regimes around submerged hydrofoils in different angle of attack (AOA), represents the major effect of environmental condition on hydrofoils performance. Thus, an appropriate prediction of 2D hydrofoil's hydrodynamic performance is a necessity to achieve an improved architecture of hydrofoils.

Force is applied to an object when fluid flows over its surface. This force that happens perpendicular to the direction of the incoming flow is defined as the drag force parallel to the lifting force. These forces are known as aerodynamic forces if the surrounding medium is air. ANSYS Fluent software was used for the numerical analysis to derive the velocity and pressure distribution on the model surface. The coefficients of lift and drag were computed using varying relative velocities. The NACA 0012 and NACA 4412 hydrofoils were subjected to a two-dimensional turbulent flow analysis investigation at varied angles of attack and velocities.

A. Binod · B. S. George (✉) · D. G. Dilip · E. P. Varghese · R. Abhishek · A. Jawaharlal
Department of Mechanical Engineering, MBCET (Autonomous), Thiruvananthapuram, Kerala, India
e-mail: bobin.george@mbcet.ac.in

K. M. Singh et al. (eds.), *Fluid Mechanics and Fluid Power, Volume 3*, Lecture Notes in Mechanical Engineering, https://doi.org/10.1007/978-981-99-6343-0_43

2 Literature Review and Objective

In order to have a submerged hydrofoil with higher controllability, Rediniotis et al. [1] developed a shape-memory-alloy actuated bio-mimetic hydrofoil. This hydrofoil shape can be deforming to different shapes mimicking aquatic animal swimming to advance its performance. Vassalos et al. [2] and Matin et al. [3] formulated a numerical procedure based on potential panel method to analyze the 3D NACA4412 hydrofoil performance under the free surface. They found the pressure distribution, lift, drag, and wave generated profile to study various conditions of the 3D hydrofoil near to the free surface. The hydro structural optimization-framework used in the work is taken from previously developed framework, MACH. The CFD solver have used ADflow, it is a 3D finite-volume, cell-centered multiblock solver for the compressible flow equations. The hydrodynamic loads (pressure and shear stresses) calculated by ADflow are given to the structural solver, and the displacements from the structural solver, in fact, dictate the CFD mesh movement. To obtain a efficient and compact set of geometric design variable [4], the FFD volume approach parameterizes the geometric variations rather than the model itself. All the geometric variations are performed on the outside boundary of the FFD volume. The gradient-based optimization algorithm, SNOPT is used to rise the computational efficiency by lowering the number of function evaluations for cases with a large number of design variables [5].

A model of study is created using software like CAD and is analyzed using ANSYS to estimate the performance of device based on lifting bodies such as rudder or Marine current turbines. In this paper it explains the formation of laminar separation bubble when it subjected under various hydrodynamic condition. By varying the conditions and different properties by using numerical formulas various results are obtained when analyzed using software like ANSYS. The laminar to turbulent operations operated by the laminar separation bubble transition. The procedure occurs when a laminar boundary layer is under a sufficiently high adverse pressure gradient that give rise to separation of the boundary layer. The Kelvin–Helmholtz instabilities are formed and increased in the various shear layer and lastly lead to a break-down of the shear layer, that results in a turbulent flow that reattaches to the model. A close bubble is formed in between the separation and reattachment points [6]. Model composite structures along with orthotropic materials uses solid elements in a framework to give the hydro elastic result of composite hydrofoils with different unidirectional orientations and to analyze the influence of the fiber orientation on the response. An interior structure is needed for hydrodynamic rising of bodies cause of the higher fluid loading. The model the composite hydrofoils uses the orthotropic material properties. Having an idea of how the free vibration results to variations in fiber orientation [7, 8]. For the design to be efficient, the hydrofoil chosen needs to have a higher value of lift and low drag properties. NACA4412 and NACA0012 have been selected for this purpose. Ansys Fluent 19.2 commercial solver is used for analysis.

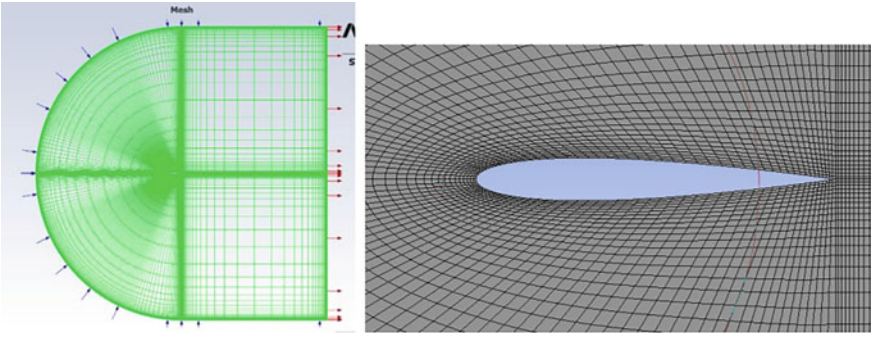

Fig. 1 Flow domain and mesh generated. The analysis has the flow around a hydrofoil at various angles of attack (4°, 8°, 12°) and varying velocities

3 Materials and Methods

The model coordinates were taken and prepared after references from the site [9], and structured mesh and un-structured mesh was created with different number of divisions and different number of elements. In order to obtain reliable resolution after trailing edge, as to attain tight cells in terms of mesh size. C-grid type is shown as structural mesh at which another node follows each of nodes consecutively. In present analysis, C type mesh with three-way velocity inlet method is used. The pressure-based implicit steady solver with realizable k-ε model turbulence model alongside second order upwind scheme is adopted for analysis. Mesh independency study was carried out and number of elements was fixed at 432,860. The domain and generated mesh is shown in Fig. 1.

4 Results and Discussion

Pressure coefficient contours at different values of attack angles were obtained for NACA0012 and NACA4412. Figure 2 depicts a general trend obtained.

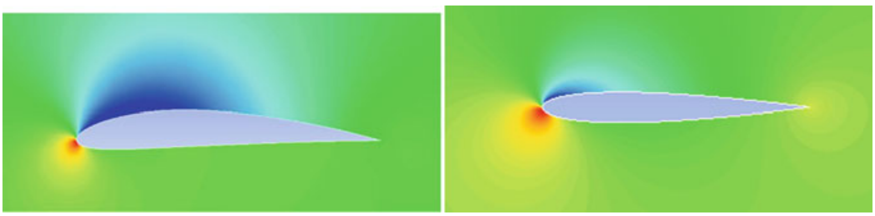

Fig. 2 Pressure contours of NACA 4412 and NACA 0012 for angle of attack 4° and velocity 5 m/s

The upper surface will be encountered by low pressure and the lower surface will be encountered by higher pressure. For this reason, the values of the lift coefficients will be rising and the values of the drag coefficients will also be rising, but the rise in drag value is less than the rise in lift values. The pressure on the lower side of each hydrofoil is higher than the pressure on the upper surface. Each hydrofoil is pushed upward more effectively into the incoming flow stream. Taking a look at the leading edge, we can see that the stagnation points where the flow velocity is almost zero for each hydrofoil. As the flow velocity accelerates over the upper surface of the hydrofoil, the velocity of the flow is completely opposite for lower surface of each hydrofoil. Figure 3 visualizes the velocity contour.

In Fig. 4, pressure coefficient distributions of the NACA0012 and NACA4412 hydrofoils at different attack angles is shown. It can be observed that the pressure coefficient varies vastly under different attack angles. Upper surface side of the hydrofoil has negative pressure coefficient and the lower surface side of the hydrofoil was positive, so the lift force of the airfoil is toward upward. When the attack angle rises, pressure coefficient difference between the lower and upper surface side is also raised. It is also found that the pressure difference coefficient is almost very lower at the trailing edge when it is much more at the leading edge. Thus, it also shows that the hydrofoil's lift force is mainly generated at the front.

Turbulent flow creates more friction drag than laminar flow due to its greater interaction with the surface side of the hydrofoil. In Fig. 5, since a turbulent boundary layer has more energy to oppose an adverse pressure gradient, it's often forces to the boundary layer to turn turbulent over fuselages to reduce overall drag. The two

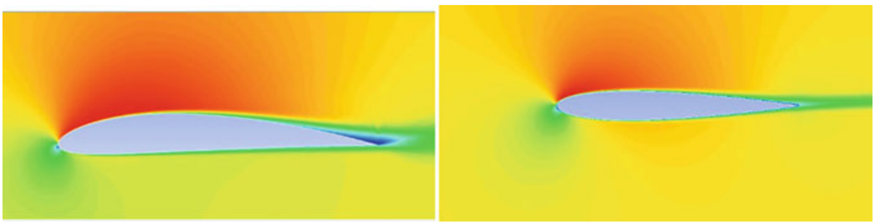

Fig. 3 Velocity contours of NACA 4412 and NACA 0012 for angle of attack 4° and velocity 5 m/s

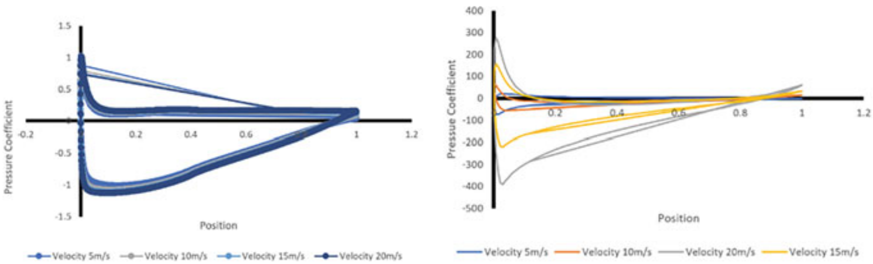

Fig. 4 Pressure coefficient values for NACA4412 and NACA0012

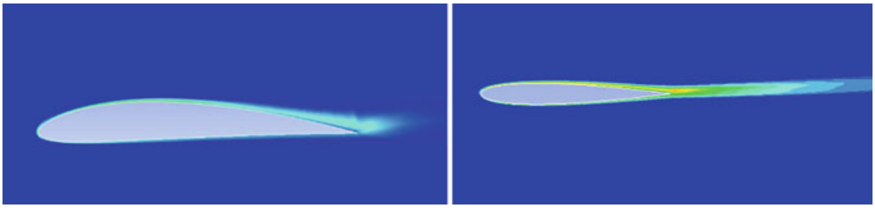

Fig. 5 Turbulence contour for NACA4412 and NACA0012 for angle of attack 4° and velocity 5 m/s

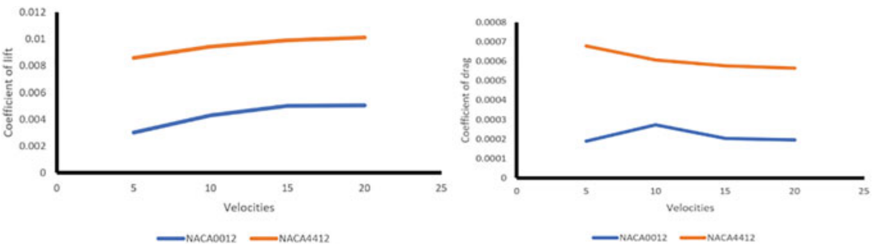

Fig. 6 Coefficient of lift and drag versus velocities

variables of turbulence effect are turbulent kinetic energy (k), it finds the energy in turbulence, and turbulent dissipation rate (epsilon), which finds the rate of dissipation of turbulent kinetic energy. So, k–epsilon model is used for having the turbulence model. With the increase in fluid free stream, the turbulence intensity causes delay of the stall angle and the maximum lift coefficient. However, it causes the increase in drag coefficient as shown in Fig. 6.

5 Conclusions

Analysis of hydrodynamic performance of NACA0012 and NACA4412 hydrofoils have been performed at different values of attack angles (4°, 8°, 12°), and using the Realizable k–ε turbulence model. The 2D analysis of the hydrofoil shaped models to navigate small and shallow waters is completed. The efficient design so that maximum performance can be obtained in an NACA0012 model. The effectiveness of the said is done by CFD. It can be seen that on the velocity magnitude figures, the upper surface side's flow velocity is higher than the lower surface's flow velocity and flow velocity of the upper surface rises with the rising of attack angles. It can be observed that from the pressure coefficient contours, pressure of the lower surface side of the hydrofoil increases with increasing attack angles. And also, the upper surface side of the hydrofoil has negative pressure coefficient and the lower surface of hydrofoil was positive, so the lift force of the hydrofoil is upwards. When the pressure coefficient contours of NACA0012 and NACA4412 are observed, it can be taken that NACA4412 will have higher pressure gradient at each angle of attack. When the velocity magnitude contours of NACA0012 and NACA4412 are observed, it can be seen that the lower surface side of the asymmetric hydrofoil (NACA4412) provides more lift than the lower surface side of symmetrical airfoil (NACA0012). When the pressure coefficient of NACA0012 and NACA4412 is examined, it is realized that area of negative pressure for NACA4412 is larger than NACA0012. From the turbulence contour it is observed that after the leading edge of NACA0012, the effect of turbulence increases, then decreases at the tailing edge. From the coefficients of lift and drag changes with the angle of attack, it can be seen that the hydrofoil NACA0012 is more efficient shape than hydrofoil NACA0012 at every attack angles. Because of the lower coefficient of drag values observed in NACA 0012, it is opted as the more self-propelling model compared to NACA4412.

References

1. Rediniotis OK, Wilson LN, Lagoudas DC, Khan MM (2002) Development of a shape-memory-alloy actuated biomimetic hydrofoil. J Intell Mater Syst Struct 13(1):35-49. https://doi.org/10.1177/1045389X02013001534
2. Vassalos D, Xie N, Jasionowski A, Lee BS (2007) Dynamic stability of the air-lifted catamaran. Proc Inst Mech Eng, Part M: J Eng Marit Environ 221(4):137–146. https://doi.org/10.1243/14750902JEME97
3. Matin RK, Ghassemi H, Arzideh MH (2016) Numerical study of the effect of geometrical changes on the airfoil aerodynamic performance. Int J Fluid Mech Res 43:28–38. https://doi.org/10.1615/InterJFluidMechRes.v43.i1.30
4. Burrows CR (2000) Fluid power systems—some research issues. Inst Mech Eng Part C: J Mech Eng Sci 214:203–220
5. Garg N et al (2019) Experimental investigation of a hydrofoil designed via hydrostructural optimization. J Fluids Struct 84:243–262
6. Delafin P-L, Deniset F, Astolfi J (2014) Effect of the laminar separation bubble induced transition on the hydrodynamic performance of a hydrofoil. Eur J Mech B/Fluids 46. https://doi.org/10.1016/j.euromechflu.2014.03.013
7. Liao Y et al (2019) Viscous fluid–structure interaction response of composite hydrofoils. Compos Struct 212:571–585
8. Panigrahi DC, Mishra DP (2014) CFD simulations for the selection of an appropriate blade profile for improving energy efficiency in axial flow mine ventilation fans. J Sustain Min 13(1):15–21
9. https://airfoiltools.com/