CFD ANALYSIS OF PRESSURE COEFFICIENT FOR NACA 4412 AT VARIOUS DEPTHS

MD. ARIFUL HOQUE

B.Sc. in Naval Architecture and Marine Engineering, BUET

Abstract - Pressure coefficient of a shallowly submerged cambered hydrofoil near free surface is simulated numerically in this research. The standard NACA 4412 hydrofoil section is used for thissimulation. A commercial computational fluid dynamics (CFD) code FLUENT based on finite volume technique is used for the analysis of the two-dimensional hydrofoil section. To get the pressure coefficient, 'Volume of Fluid' (VOF) method is incorporated in numerical simulation. Then pressure coefficients are computed for different submergence depth ratios, h/c ranging from 1 to 5. Variation of pressure coefficientsis plotted in the form of a dotted curve for F_n=1, and R_e=3.11x 10^6in different h/c ratios.

Keywords - Pressure Coefficients, Drag coefficient, Liftcoefficient, Hydrofoil, Computational domain.

I. INTRODUCTION

In a 2-D flow, it is necessary to assume a hydrofoil to be a wing of infinite span. A real wing starts with a hub and ends with a tip. In case of a real wing the chord and twist change along the span According to Prandtl, local 2-D data for the forces can be used if the angle of attack is corrected accordingly with the trailing vortices behind the wing.2-D hydrodynamics is of practical interest even though it is difficult to realize. For the analysis, the two-dimensional model of the hydrofoil and the mesh is created through Point-wise which is run in Fluent for numerical iterate solution.

The k-ε turbulence model has been implemented to simulate turbulent flow past the foilin the free surface zone at different submergence ratios (h/c). Fig 1 shows the leading edge stagnation point present in the 2-D flow past a hydrofoil.

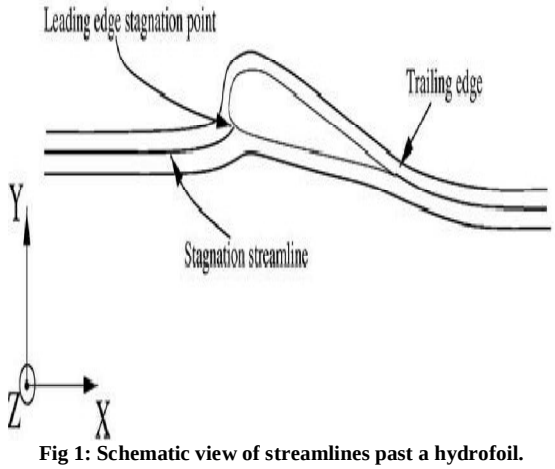


Fig 1: Schematic view of streamlines past a hydrofoil.

Fig 2 shows the leading edge stagnation point present in the 2-D flow past a hydrofoil. The reacting force F from the flow is divided into a direction perpendicular to the velocity at infinity V∝ and to a direction parallel to V∝. The former component is known as the lift, L; the latter is called a drag, D.

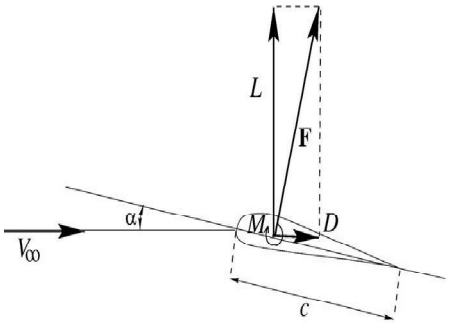


Fig 2: Definition of lift and drag

Nomenclature	
c	chord length
h	distance from the free surface
h/c	submergence ratio
F_n	Froude Number
R_n	Reynolds number

II. MATHEMATICAL FORMULATION

2.1 Navier-Stokes equations

The Navier-Stokes equations are the basic governing equations for a viscous, compressible real fluid. It is a vector equation obtained by applying Newton's Law of Motion to a fluid element and is also called the momentum equation. It is supplemented by the mass conservation equation also called continuity equation. The instantaneous continuity equation and momentum equation for a compressible fluid can be written as:

$$\partial p/\partial t + \nabla.(\rho.u) = 0 \quad (1)$$

$$\rho(\partial u/\partial t + u.\nabla u) = -\nabla p+\nabla.(\mu(\nabla u+(\nabla u)^T))+\nabla((-2\mu/3)\nabla.u)+\rho g \quad (2)$$

2.2Volume of Fluid (VOF) Model

The VOF formulation relies on the fact that two or more fluids (or phases) are not interpenetrating. For each additional phase that is added to the model, a variable is introduced: the volume fraction of the

phase in the computational cell. In each control volume, the volume fractions of all phases sum to unity. The fields for all variables and properties are shared by the phases and represent volume-averaged values, as long as the volume fraction of each of the phases is known at each location. Thus the variables and properties in any given cell are either purely representative of one of the phases, or representative of a mixture of the phases, depending upon the volume fraction values. In other words, if the q^{th} fluid's volume fraction in the cell is denoted as α_q, then the following three conditions are possible:

- $\alpha_q = 0$: the cell is empty (of the q^{th}fluid).
- $\alpha_q = 1$: the cell is full (of the q^{th}fluid)
- $0 < \alpha_q < 1$: the cell contains the interface between the q^{th}fluid and one or more other fluids.

Based on the local value of α_q , the appropriate properties and variables will be assigned to each control volume within the domain.

The tracking of the interface(s) between the phases is accomplished by the solution of a continuity equation for the volume fraction of one (or more) of the phases. For the q^{th}phase, this equation has the following form:

$$\frac{1}{\rho_q}\left[\frac{\delta}{\delta t}(\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \overrightarrow{v_q}) = S_{\alpha_q} + \sum_{p=1}^{n}\left(\dot{m}_{pq} - \dot{m}_{qp}\right)\right] \quad (3)$$

Where, $\dot{m}_{qp}$ is the mass transfer from phase q to phase p and $\dot{m}_{pq}$ is the mass transfer from phase p to phase q. By default, the source term on the right-hand side of above equation, $S_{\alpha q}$, is zero, but you can specify a constant or user-defined mass source for each phase.

The volume fraction equation will not be solved for the primary phase; the primary-phase volume fraction will be computed based on the following constraint:

$$\sum_{q=1}^{n} \alpha_q = 1 \quad (4)$$

2.3 Simulation Using Fluent

FLUENT uses a finite volume based algorithm to transform the governing physical equations to algebraic equations that can be solved numerically. In such an approach, the computational domain is subdivided into individual, discrete control volume, or cells. The governing equations about each cell are then integrated, yielding discrete equations that conserve each quantity on a control-volume basis. Let us consider the following steady-state conservation equation for transport of a scalar quantity ϕ written in integral form for an arbitrary control volume V:

$$\int \rho\phi \vec{v}.d\vec{A} = \int \Gamma_\Phi \vec{\nabla}\Phi.d\vec{A} + \int S_\Phi dV \quad (5)$$

Where ρ is the density, v is the velocity vector (= ui+vj in 2D), A is the surface area vector, Γ_Φ is the diffusion coefficient for ϕ, $\vec{\nabla}\Phi$ is the gradient of ϕ and S_ϕ is the source of ϕ per unit volume.

This equation is applied to each cell in the computational domain. Fluent discretizes this integral equation as:

$$\sum_{f}^{N_f} \overrightarrow{v_f}\Phi_f A_f - \sum_{f}^{N_f} \Gamma_\Phi \left(\vec{\nabla}\Phi\right)_n A_f + S_\Phi$$

where N_f is the number of face enclosing cell,

$\overrightarrow{v_f}$ is the mass flux through the face,

A_f is the area of face f,

$\left(\vec{\nabla}\Phi\right)_n$ is the magnitude of $\vec{\nabla}\Phi$ normal to face f and V is the cell volume.

III. COMPUTATIONAL DOMAIN

The flow field around the hydrofoil is modeled in two dimensions. The flow from left to right with the hydrofoil of chord length c=3.825 m submerged in an incompressible fluid (seawater) is considered, as the experiment was also carried out by taking the same length of the hydrofoil. The total 2D computational domain with different boundary conditions (i.e., 'velocityinlet'applied to upstream, upper, lower boundaries and 'pressure outlet' to downstream)are shown in Figure 4.

The length of the inlet and outlet boundaries is 9c each, and upper and lower boundaries are 11c each where c is the chord length of the hydrofoil. The hydrofoil is positioned at a distance 3c right from the inlet boundary and 4c down from the upper boundary. The origin of the coordinate system is placed at the leading edge of the hydrofoil. The hydrofoil is inclined at an angle of 5^0 since the experiment was carried out by considering an angle of attack of 5^0.

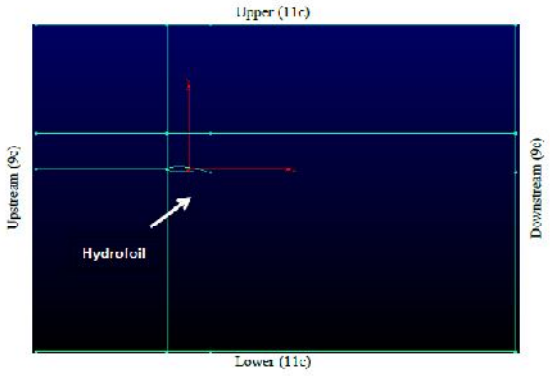


Figure 4: Computational domain with different boundaries.

The different boundaries with boundary conditions are shown in Table 1.

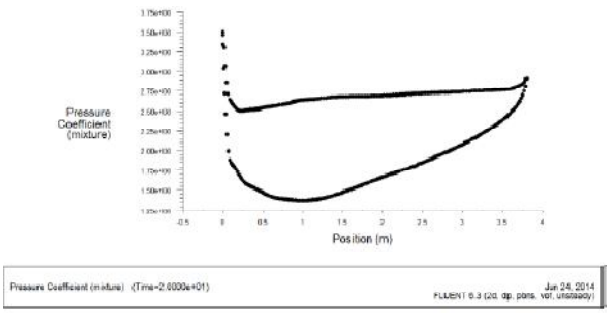
Table 1: Boundary Condition

Boundary	Boundary type
Upstream (Inlet)	Velocity Inlet
Downstream (Outlet)	Pressure Outlet
Upper	Velocity Inlet
Lower	Velocity Inlet
Hydrofoil	Wall (no slip)

IV. RESULTS AND DISCUSSION

The pressure coefficient of the lower surface is always higher than pressure coefficient onthe upper surface. In the lower surface it remains almost constant but in the upper surface rises gradually from the leading edge to the trailing edge of hydrofoil which is shown in Figures 5-7.The values of pressure coefficient on the trailing edge are increased which is shown in table 2.

a



b

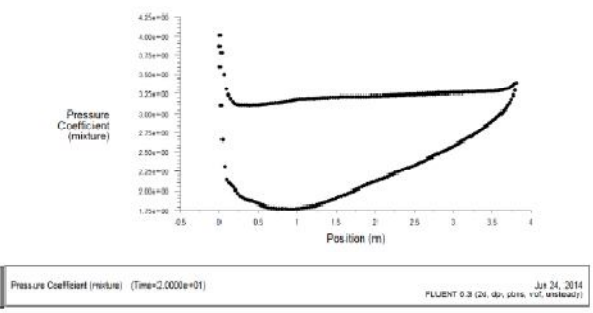


Figure 5(a) Pressure coefficient around NACA 4412 hydrofoil at h/c=1, (b)Pressure coefficient around NACA 4412 hydrofoil at h/c=1.5

b

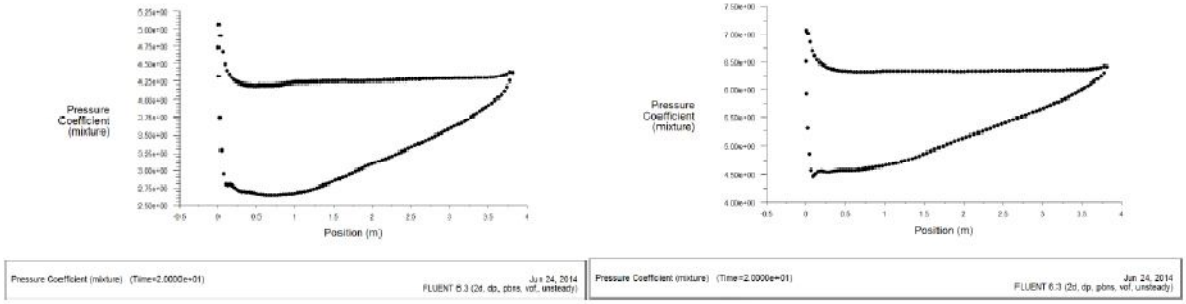


Figure 6(a) Pressure coefficient around NACA 4412 hydrofoil at h/c=2, (b)Pressure coefficient around NACA 4412 hydrofoil at h/c=3

a

b

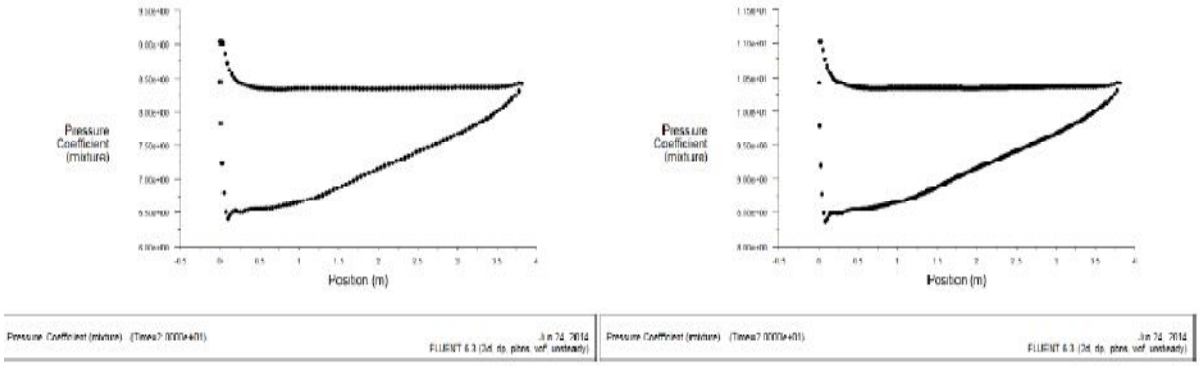


Figure 7(a) Pressure coefficient around NACA 4412 hydrofoil at h/c=4, (b)Pressure coefficient around NACA 4412 hydrofoil at h/c=5

Table 2: Pressure Coefficient at Trailing Edge

Submergence depth	Pressure coefficient at trailing edge
1.0c	2.40
1.25c	2.95
1.5c	3.40
2.0c	4.35
2.5c	5.40
3.0c	6.45
4.0c	8.45
5.0c	10.45

From Table 2 it is seen that with the increase in submergence depth, the pressure coefficient at the trailing edge of NACA 4412 hydrofoil is increased gradually.

4.1Values of C_L and C_D at different submergence depth

The lift and drag coefficient of the hydrofoil at an angle of attack 5^0 for different submergence depth ratios are shown in Table 3.

Submergence Depth Ratio, h/c	Lift Coefficient, C_L	Drag Coefficient, C_D
1.00	0.76368	0.06805
1.25	0.73969	0.05494
1.50	0.72109	0.04584
2.00	0.71440	0.03246
2.50	0.71003	0.02292
3.00	0.70664	0.01728
4.00	0.69574	0.01413
5.00	0.69072	0.01202

CONCLUSION

1. To analyse flow characteristics around the cambered asymmetric hydrofoil NACA 4412, the Finite Volume Methodis quiet successful.
2. The interface between air and water, known as free surface can be captured precisely by the volume of fluid (VOF) method.
3. The values of lift coefficients and drag coefficients of hydrofoil gradually decrease with the increase of submergence depth.
4. With the increase in submergence depth, the pressure coefficient at the trailing edge of NACA 4412 hydrofoil is increased gradually.
5. The Pressure coefficient on the lower surface of the hydrofoil was greater than that of the incoming flow stream and as a result, the hydrofoil was pushed upward; normal to the incoming flow stream.

REFERENCES

[1] Abott IH, Von Doenhoff AE. Theory of wing sections. ISBN 486-60586-8 Dover Publishing, New York.(1959)

[2] Klaus A. Hoffmann, Steve T. Chiang "Computational Fluid Dynamics" Volume I, Fourth edition (August 2000)

[3] Bai, K.J. and Han, J.H. (1994). A localized finite-element method for the nonlinear steady waves due to a two-dimensional hydrofoil. J. Ship Res. 38, 42–51.

[4] Bal, S. (2008). Prediction of wave pattern and wave resistance of surface piercing bodies by a boundary element method. Int. J. Numer. Meth. Fluids, 56(3), 305-329.

[5] Dawson, C. W. (1977). A practical computer method for solving ship-wave problems. Proceedings of Second International Conference on Numerical Ship Hydrodynamics, 30-38.

[6] Ghassemi, H.andKohansal, M.R. (2010) Iso-parametric boundary element method applied to the rectangular and delta hydrofoils near free surface, China Ocean eng., 24(2)

[7] Janson, C.E. (1997). Potential flow panel method for the calculation of free surface flows with lift, Ph. D. thesis,Chalmers University of Technology.

[8] Kouh J. S. (2002). Performance analysis of two dimensional hydrofoil under free surface.Journal of National Taiwan University 86.

[9] Xie, N. and Vassalos, D. (2007).Performance analysis of 3D hydrofoil under free surface. Ocean Engineering, 34, 1257–1264.

[10] Yeung, R. W. andBouger, Y. C. (1979). A hybrid-integral equation method for steady two-dimensional ship waves. Int. J. Num. Meth. Eng., 14, 317-336

[11] H. Ghassemi and A. R. Kohansal. (October 28, 2011). "Wave Generated by the NACA4412 Hydrofoil near Free Surface".

[12] Hirt, C.W Nicholas,B.D (1981), "Volume of fluid (VOF) method for the dynamics of free boundaries", Journal of computational physics 39(1):201-225

[13] Shyy,W.,(1997), " Computational modeling for fluid flow and interfacial transport", Elsevier, Amsterdam.

[14] Spalding D., "Numerical computation of multiphase flows",VKI for Fluid Dynamics. Brussels,1981

★★★