Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 194 (2017) 38 – 43

Procedia Engineering

www.elsevier.com/locate/procedia

10th International Conference on Marine Technology, MARTEC 2016

Simulation of Water Wave Generated by Shallowly Submerged Asymmetric Hydrofoil

Md. Ariful Hoque[a,*], Md. Mashud Karim[b], Aevelina Rahman[c]

[a]*Abul Khair Steel Mills Ltd., Chittagong, Bangladesh*
[b]*Dept. of Naval Architecture and Marine Engineering,BUET,Dhaka-1000*
[c]*Graduate Student, Dept. of Mechanical Engineering, Virginia Tech., VA, USA*

Abstract

In this study, the wave generated by flow around a shallowly submerged cambered hydrofoil near free surface is simulated numerically. The standard NACA 4412 hydrofoil section is used for ease of comparison with available experimental data. A commercial computational fluid dynamics (CFD) code FLUENT based on finite volume technique is used for the analysis of the two dimensional hydrofoil section. The k-ϵ turbulence model has been implemented to simulate turbulent flow past the foil in the free surface zone at different submergence ratios (h/c). To get the free surface elevation, 'Volume of Fluid' (VOF) method is incorporated in numerical simulation. To validate the computational results, the free surface wave generated by the flow around hydrofoil at submergence depth ratio h/c = 1 is compared with experimental results. The computed results show satisfactory agreement with the experimental measurements. Then free surface effects on wave profile are computed for six submergence depth ratios, h/c ranging from 1 to 5. Finally, the pressure coefficients at the trailing edge and hydrodynamic forces are computed at F_n=1, R_e=3.11 × 10^6 for different h/c ratios.

Peer-review under responsibility of the organizing committee of the 10th International Conference on Marine Technology.
Keywords: Free surface effect; hydrofoil; CFD; viscous flow; Volume of Fluid (VOF) method.

1. Introduction

The wave generation due to the presence of a body moving at steady forward speed beneath a free surface has been the subject of extensive research work[1–6] in marine hydrodynamics. In this study the wave generated by flow around a shallowly submerged cambered hydrofoil near free surface is simulated numerically. The standard NACA 4412 hydrofoil section is used for ease of comparison with available experimental data. A commercial computational fluid dynamic (CFD) code ANSYS FLUENT based on finite volume technique[7] is used for the analysis and 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 foil in the free surface zone at different submergence ratios (h/c). To get the free surface elevation, "Volume of Fluid" (VOF) method[8] is incorporated in numerical simulation. To validate the computational results, the free surface wave generated by the flow around hydrofoil at submergence depth ratio h/c = 1 is compared with experimental results. The computed results show satisfactory agreement with the experimental measurements. Finally, free surface effects on

* Corresponding author. Tel.:+880191467604
E-mail address: arifhoque777@gmail.com

Peer-review under responsibility of the organizing committee of the 10th International Conference on Marine Technology.
doi:10.1016/j.proeng.2017.08.114

wave profile are computed for six submergence depth ratios, h/c ranging from1 to 6. Then the wave profiles, contours of velocity magnitude , static pressure near the hydrofoil and free surface are computed at F_n=1, R_e=3.11 × 10^6 for different h/c ratios.

Nomenclature

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

2. 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:

$$\frac{\partial p}{\partial t} + \nabla.(\rho.u) = 0 \tag{1}$$

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

2.2. Volume 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 th fluid).
- $\alpha_q = 1$: the cell is full (of the th fluid)
- $0 < \alpha_q < 1$: the cell contains the interface between the 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:

$$\frac{1}{\rho_q}[\frac{\delta}{\delta t}(\alpha_q\rho_q) + \nabla.(\alpha_q\rho_q\vec{v_q}) + \sum_{p=1}^{n}(\dot{m}_{pq} - \dot{m}_{qp}] \tag{3}$$

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