International Journal of Heat and Mass Transfer 83 (2015) 591–603

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Numerical study of cloud cavitation effects on hydrophobic hydrofoils

Jonghyun Kim, Joon Sang Lee [*]

Department of Mechanical Engineering, Yonsei University, Seoul 120-749, Republic of Korea

ARTICLE INFO

Article history:
Received 19 June 2014
Received in revised form 17 December 2014
Accepted 19 December 2014
Available online 8 January 2015

Keywords:
Cloud cavitation
Phase change
Large eddy simulation
Mass transfer model
Partial slip boundary condition
Hydrofoil

ABSTRACT

To explore the effect of hydrophobicity on cloud cavitation, the behaviors of cloud cavitation over the Clark-Y hydrofoil under various slip condition were investigated. Large eddy simulation (LES) was used for the turbulence model. The mass transfer model, which was considered to be a two-phase mixture flow, was used for the vaporization and condensation processes in the transport equation. The volume of fluid (VOF) scheme was used to track the interface of the dispersed phase by using the local volume fraction. Slip strength was controlled using the friction coefficient. The cavitation model in this study agreed with experimental and previous numerical studies. The results show that as the slip strength grew stronger, the friction drag was reduced; the cavity became longer and the shedding frequency decreased. For this reason, cloud cavitation is stabilized in condition of strong slip strength. Thus, a relatively weak re-entrant jet occurs in conditions of strong slip strength which gives rise to small amount of vapor shedding at the closure. This means that cloud cavitation instability was alleviated as the hydrophobicity increased.

1. Introduction

Cavitation is a general phenomenon of fluid mechanics that is generated from pressure and velocity fluctuations in the fluid. It usually occurs when the pressure in a certain area drops below the vapor pressure. Cavitation can appear in a wide variety of propulsion systems such as pumps, marine propellers and hydrofoils. Many investigators have studied numerous aspects of cavitation over the last several decades [1,2]. However, cavitating flow, including turbulence and multi-phase flow with phase change, is complex and unsteady. Hence, the mechanism behind cavitating flow has not been yet fully understood.

For a given Reynolds number and incidence angle, incipient cavitation, sheet cavitation, cloud cavitation and super-cavitation appear in response to the cavitation number [3]. Above all, undesirable effects caused by the violent and catastrophic collapse of cavitation bubbles in cloud cavitation are more serious than the effects of other cavitation regimes, and result in the production of noise as well as the possibility of material damage to nearby solid surfaces [1]. Accordingly, cloud cavitation behaviors are considered an important part in cavitation instability.

To model cavitation phenomena, multiphase flows and the dynamics of change between the two phases need to be considered. For these, it is important to choose the appropriate mass transfer model, also called the cavitation model. Over the past few decades, a number of cavitation models have been developed by many investigators [4–8]. According to Senocak et al. [9], the Kunz model accurately predicts the cavities. Roohi et al. [10] compared the Kunz model with the Sauer model and showed that the Kunz model is superior to the Sauer model for cloud cavitation. Therefore, we employed the Kunz model.

It is widely known that the important factors influencing cavitation development around a hydrofoil are fluid conditions, foil shape and surface characteristics. While most of the cavitation studies have focused on the influence of fluid conditions [2,3,11] and foil shape [12–14], studies on the influence of surface characteristics are rather scarce. In previous cavitation studies, the hydrophobicity of a solid surface was generally thought to play an important role in cavitation development [15]. Leger et al. [16] observed cavitating flows on smooth surfaces with both hydrophilic and hydrophobic conditions. They showed that for the hydrophobic surface the distance between the position of the boundary layer separation and the cavity detachment became smaller as the Reynolds number grew larger. Bremond et al. [17] performed an investigation on the presence of gas on flat and smooth hydrophobic surfaces lying underwater. They determined that the nucleation of bubbles takes place on an initially smooth surface for much smaller negative pressure than the pressure threshold of water rupture. Kawakami et al. [18] revealed remarkably different cavity shedding appearances and behaviors for a cavitating NACA0015 foil between three different water tunnels. They argued that surface effects could have

---

[*] Corresponding author. Tel.: +82 2 2123 5820; fax: +82 2 312 2159.
E-mail address: joonlee@yonsei.ac.kr (J.S. Lee).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2014.12.051
0017-9310/

a significant influence on the fully wetted time during cavity shedding. However, the results of most research conducted so far do not agree and the mechanism of cavitation remains an open question. Although there has been significant interest in the use of hydrophobic surfaces for cavitation studies, insufficient numerical studies have been conducted.

The objective of this study is to explore the behavior of cloud cavitation on hydrophobic hydrofoils and to establish the dependence on the slip strength as illustrated in Fig. 1. The cloud cavitating flow around a Clark-Y hydrofoil at a specific incidence angle and cavitation number was investigated. For hydrophobic surfaces, a slip boundary condition on the hydrofoil surface was applied. It has been proven by many investigators that liquid–solid interactions vary differently depending on the degree of hydrophobicity [19]. Thus, we compared the behaviors of cavitation under differing slip strength conditions. Simulation results, including cavitating structures and dynamic characteristics were used to study the influence of hydrophobic surface on cloud cavitating flow.

In this paper, we used an incompressible large eddy simulation (LES). The flow was considered to be single fluid and multiphase mixture. The volume of fluid (VOF) method was used to investigate the phase interface, and the mass transfer model was used to represent the vaporization and condensation processes in the transport equation. The governing equation of the model was discretized by a cell-centered finite volume method. In the following sections, we describe the mathematical model, followed by the numerical methods and the solution strategy including the computational domain, boundary conditions and specific properties. For model validation, the verification test was reported. The numerical results were described, followed by the concluding remarks.

2. Computational methods

2.1. Large eddy simulation (LES)

In LES, the large unsteady turbulent motions are directly represented, whereas the effects of smaller scale motions are modeled [20]. In comparison with the Reynolds-averaged Navier–Stokes (RANS) method, transient flow structures are naturally and consistently formed in LES. When modeling cavitation phenomena, this characteristic of LES is significant in order to properly predict mechanisms of cavitation dynamics such as the formation process and shedding. For incompressible flow, the governing equation of LES can be expressed as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \bar{\boldsymbol{v}}) = 0, \tag{1}$$

$$\frac{\partial (\rho \bar{\boldsymbol{v}})}{\partial t} + \nabla \cdot (\rho \bar{\boldsymbol{v}} \times \bar{\boldsymbol{v}}) = -\nabla \bar{p} + \nabla \cdot (\bar{\boldsymbol{\tau}} - \boldsymbol{\sigma}),$$

where the over-bar ($^-$) denotes filtered components. $\boldsymbol{\sigma}$ $(= 2\mu_t \boldsymbol{S})$ is the viscous stress tensor, where the rate of strain tensor is expressed as $\boldsymbol{S} = \frac{1}{2}(\nabla v + \nabla \boldsymbol{v}^T)$, and μ_t is the subgrid scale (SGS) eddy viscosity. In this study, the SGS eddy viscosity is defined as $\mu_t = C\Delta k^{1/2}$, where C is a model constant and k is the subgrid kinetic energy, and is called the "one equation SGS" model.

2.2. Multiphase flow modeling

In this study, we considered a "two-phase mixture" approach that uses a local vapor volume fraction and a transport equation with source terms for the mass transfer rate between the two phases:

$$\frac{\partial \gamma}{\partial t} + \nabla \cdot (\gamma \boldsymbol{v}) = \dot{m}. \tag{2}$$

Density and dynamic viscosities of the mixture are expressed by the vapor volume fraction, γ:

$$\rho = \gamma \rho_v + (1 - \gamma)\rho_l, \tag{3}$$

$$\mu = \gamma \mu_v + (1 - \gamma)\mu_l,$$

where ρ is density, μ is viscosity and the subscripts v and l indicate "vapor" and "liquid", respectively.

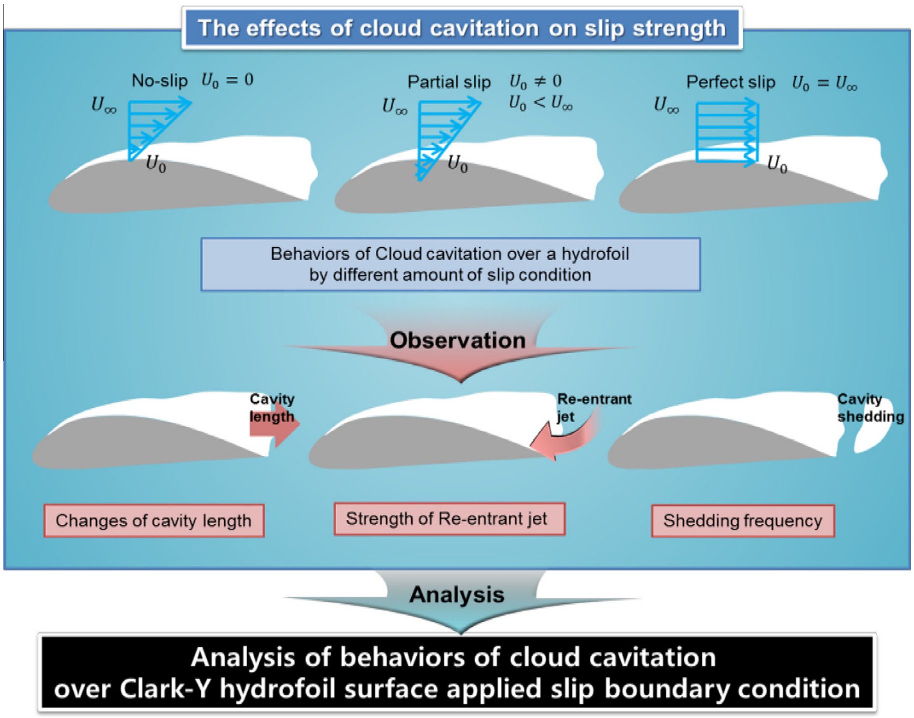

Fig. 1. A schematic of the objective of this study.