Applied Mathematical Modelling 37 (2013) 6469–6488

Contents lists available at SciVerse ScienceDirect

Applied Mathematical Modelling

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

Numerical simulation of cavitation around a two-dimensional hydrofoil using VOF method and LES turbulence model

Ehsan Roohi *, Amir Pouyan Zahiri, Mahmood Passandideh-Fard

Department of Mechanical Engineering, Faculty of Engineering, Ferdowsi University of Mashhad, P.O. Box 91775-1111, Mashhad, Iran

ARTICLE INFO

Article history:
Received 3 May 2012
Received in revised form 16 July 2012
Accepted 7 September 2012
Available online 18 September 2012

Keywords:
Clark-Y hydrofoil
Cloud cavitation
Supercavitation
LES
VOF
Mass transfer model

ABSTRACT

In this paper simulation of cavitating flow over the Clark-Y hydrofoil is reported using the large eddy simulation (LES) turbulence model and volume of fluid (VOF) technique. We applied an incompressible LES modelling approach based on an implicit method for the subgrid terms. To apply the cavitation model, the flow has been considered as a single fluid, two-phase mixture. A transport equation model for the local volume fraction of vapour is solved and a finite rate mass transfer model is used for the vapourization and condensation processes. A compressive volume of fluid (VOF) method is applied to track the interface of liquid and vapour phases. This simulation is performed using a finite volume, two phase solver available in the framework of the OpenFOAM (Open Field Operation and Manipulation) software package. Simulation is performed for the cloud and super-cavitation regimes, i.e., σ = 0.8, 0.4, 0.28. We compared the results of two different mass transfer models, namely Kunz and Sauer models. The results of our simulation are compared for cavitation dynamics, starting point of cavitation, cavity's diameter and force coefficients with the experimental data, where available. For both of steady state and transient conditions, suitable accuracy has been observed for cavitation dynamics and force coefficients.

1. Introduction

Formation of vapour bubbles within a liquid when its pressure is less than the saturated vapour pressure is called cavitation. Cavitation usually could appear over marine vehicles such as marine propeller blades. For efficiency reasons, the propeller usually needs to operate in cavitating conditions but the negative effects of cavitation such as vibrations, noise and erosion should be avoided. The radial section of these marine blades is a two-dimensional hydrofoil. Cavitation process is characterized by a dimensionless number; i.e., $\sigma = \frac{P_\infty - P_v}{0.5\rho U_\infty^2}$ called cavitation number, where p_v is the vapour pressure, ρ is the liquid density, and P_∞, U_∞ are the free stream flow pressure and velocity, respectively. Five different cavitation regimes are observed in the flow over a body: incipient, shear, cloud, partial, and supercavitation. Partial and cloud cavitation regimes refer to the situation where vapour phase covers a subsection of the body. Alternatively, supercavitation refers to a long cavity that extends more than the body length and closes in the liquid. In all cavitation regimes, there is a constant movement of a re-entrant liquid jet in the cavity closure section. In cloud cavitation regime, this backward movement of liquid results in detachment of large vapour sections from the main body [1].

Numerical simulation of cavitating flows had shown a rapid progress during the last two decades. The key challenges in numerical modelling of cavitating flows include sharp changes in the fluid density, existence of a moving boundary and the requirement of modelling phase change. Among different cavitation models, ''homogeneous equilibrium flow model'' had

* Corresponding author. Tel.: +98 (511) 8805136; fax: +98 (511) 8763304.
E-mail address: e.roohi@ferdowsi.um.ac.ir (E. Roohi).

0307-904X/$ - see front matter
http://dx.doi.org/10.1016/j.apm.2012.09.002

been widely employed [2]. This approach defines a single-fluid model for both phases. Various categories in ‘‘homogeneous equilibrium flow model’’ differ in the relation that defines the variable density field. A barotropic water-vapour state law could be applied to evaluate density field. However, selection of an appropriate state law is a difficult task and needs enough experimental data for any specific problem. Moreover, a typical barotropic state equation neglects vorticity production at the cavity closure [3]. A more appropriate approach is to solve an advection equation for liquid or vapour volume fraction and compute density as a weighted average of the volume fraction of the two phases. This approach, namely ‘‘Transport Based Equation Model (TEM)’’, has extensively been used to simulate cavitating flows. The convective characteristic of the advection equation considers the effects of inertial forces like bubbles shedding from cavities [2]. Three key points should be considered regarding the TEM approach: (1) selection of an appropriate mass transfer model, (2) a solution strategy for the advection equation, (3) appropriate turbulence model.

Sauer [4] and Yuan et al. [5] suggested cavitation models based on the classical Rayleigh equation with some improvements. Singhal et al. [6], Merkle et al. [7] and Kunz et al. [8] suggested alternative mass transfer models based on semi-analytical equations. Senocak and Shyy [9] developed an analytical cavitation model based on the mass-momentum balance around the cavity interface.

Volume of fluid (VOF) technique could be utilized to solve the advection equation of the volume fraction and predict the cavity interface accurately. The VOF technique is famous for its application in numerical simulation of free surface flows, e.g. drop collision, liquid sloshing, fluid jetting, and spray deposition [10]. This method is conservative, robust and capable of treating both of large deformations of interface as well as small-scale interface topologies such as breakup and reconnection. Consistently, the cavity interface can be tracked by VOF approach. Different VOF methods for tracking free surface interface have been developed; the most known are ‘‘Simple Line Interface Calculation’’ (SLIC) [11], Hirt–Nichols [12], ‘‘Piecewise Linear Interface Calculation’’ (PLIC) [13], and ‘‘Compressive Interface Capturing Scheme for Arbitrary Meshes’’ (CICSAM) [14,15]. In SLIC and Hirt–Nichols approaches, the interface is reconstructed with piecewise constant and piecewise constant stair-stepped line segments, respectively. However, in the PLIC method, piecewise linear segments are used to reconstruct the interface. In contrast to the geometric reconstruction algorithms [11–13], compressive scheme benefits from a high resolution differencing schemes to calculate volume fluxes [14]. Additionally, the implementation of compressive algorithms on arbitrary unstructured meshes is quite straightforward. A review of the literature shows that VOF method is in accordance with cavitation physics and can capture the cavity shape accurately. For example, Frobenius and Schilling [16], Wiesche [17], and Bouziad et al. [18] used VOF technique to simulate cavitation over hydrofoils and pump impellers.

Since most of the cavitating flows performs at high Reynolds number and under unsteady condition, implementation of a suitable turbulence model is of great importance for accurate prediction of cavitation. Different approaches such as standard or modified two-equation turbulence models (k–ε, k–ω) have been utilized to implement turbulence effects on cavitating flows [19–23]. Use of ‘‘large eddy simulation (LES)’’ is another approach considered recently in numerical cavitation modelling [24–27]. LES resolves large scales energy-containing eddies while it models small scale energy-dissipative one. The success of the LES approach in capturing the details of small-scale flow structures in cavitating flows demonstrates the important role of turbulence modelling in the cavitation prediction.

As a continuation of our previous work [28], in this study we utilize a multi-phase flow solver of OpenFOAM package to simulate cloud and supercavitation regime over two-dimensional Clark-Y hydrofoil whose experimental data is available [2]. Our simulation employs a compressive VOF technique [15] combined with two mass transfer models, namely Sauer model [4] and Kunz model [8]. VOF model used in this work considers the effect of the surface tension force over the cavity surface. Moreover, in order to capture unsteady features of cavitating flow accurately, we use an implicit large eddy simulation (LES) turbulence approach. PISO (pressure implicit with splitting of operators) algorithm is used to solve the set of governing equations [29]. The results of our simulation are compared with the experimental data for cavitation dynamics, starting point of cavitation and cavity diameter as well as lift and drag coefficients. Additionally, a comparison between two standard cavitation models, i.e., Sauer and Kunz models, and between the LES and standard k–ε turbulence model, will be reported.

2. Governing equations

2.1. Implicit LES model

Large eddy simulation (LES) is based on computing the large, energy-containing eddy structures which are resolved on the computational grid, whereas the smaller, more isotropic, subgrid structures are modelled. Development of the LES encounters a main obstacle of the strong coupling between subgrid scale (SGS) modelling and the truncation error of the numerical discretization scheme. This link could be exploited by developing discretization methods where the truncation error itself acts as an implicit SGS model. Therefore, the ‘‘implicit LES’’ expression is used to indicate approaches that merge SGS model and numerical discretization [30]. Furthermore, the cell-averaging discretization of the flow variables can be thought of as an implicit filter. In the other words, finite volume discretization provides top-hat-shaped-kernel filtered values as:

$$\bar{f}_p = \frac{1}{(\delta V_p)} \int_{\Omega_p} f dV, \tag{1}$$