Ocean Engineering 112 (2016) 287–306

Contents lists available at ScienceDirect

Ocean Engineering

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

Investigation of cavitation around 3D hemispherical head-form body and conical cavitators using different turbulence and cavitation models

Mohammad-Reza Pendar, Ehsan Roohi *

High Performance Computing (HPC) Laboratory, Department of Mechanical Engineering, Faculty of Engineering, Ferdowsi University of Mashhad, P.O. Box 91775-1111, Mashhad, Iran

ARTICLE INFO

Article history:
Received 18 January 2015
Accepted 7 December 2015
Available online 31 December 2015

Keywords:
Conical cavitator
Hemispherical head-form body
Supercavitation
Volume-of-Fluid (VOF) method
OpenFOAM

ABSTRACT

In this paper, cavitation and supercavitation around 3D hemispherical head-form body and a conical cavitator were simulated. Dynamic and unsteady behaviors of cavitation were solved using large eddy simulation (LES) and k-ω SST turbulence models, as well as Kunz and Sauer mass transfer models. In addition, the compressive volume of fluid (VOF) method is used to track the cavity interface. Simulation is performed under the framework of the OpenFOAM package. The main contribution of this work is to present a correlation between the cavity length and diameter for hemispherical head-form bodies for the first time. Moreover, we provide a detailed comparison between different turbulence and mass transfer models over a broad range of cavitation numbers, especially in small cavitation numbers, including σ=0.07, 0.05, 0.02 for two cases, which is not reported previously. Our numerical results are compared with the available experimental data and a broad set of analytic relations for the cavity characteristics such as cavity length and diameter with suitable accuracy. Discussions on boundary layer separation and re-entrant jet behavior, which play a significant role in the bubble shedding in the cavity closure region, are presented.

1. Introduction

Cavitation is a multi-phase and complex physical phenomenon defined as the formation of vapor bubbles within a liquid when the pressure locally drops below the saturated vapor pressure (Brennen, 1995). As a useful phenomenon, cavitation attracted the attention of many researchers in the past decades. Cavitation usually appears over underwater vehicles such as underwater vehicles, submarine, hydrofoils and marine propeller blades. Formation of cavity cloud significantly increases the performance of the marines by reducing the viscous drag. Cavitation is a three-dimensional and periodic phenomenon that exhibits unsteady dynamic behaviors such as periodic shedding of the cavity cloud and growth and rapid collapse of bubbles (Wang and Ostoja, 2007). Cavitation could be categorized by a dimensionless number; i.e., $\sigma=(P_\infty - P_\vartheta)/0.5\rho U_\infty^2$ that is called cavitation number. When one decreases the cavitation number, i.e., via increasing the velocity of the moving body further, supercavitation will occur which consists of a long and steady cavity region.

Precise simulation of cavitation phenomenon requires an accurate mass transfer model, a surface reconstruction scheme for capturing the sharp cavity interface as well a suitable turbulence model. Different kinds of mass transfer model can be used for cavitation modeling. Famous cavitation models based on semi-analytical approaches were derived by Merkle et al. (1998), Kunz et al. (2000), Sauer (2000); Yuan et al. (2001) and Singhal et al. (2002).

Among different surface reconstruction schemes, Volume of Fluids (VOF) method was extensively used to describe the phase transition mechanism between liquid and vapor phases that both exist in cavitating flows. For instance, Passandideh Fard and Roohi (2008), Shang, (2013), Roohi et al. (2013), Yu et al. (2014) and Kim and Lee (2015) used VOF method to simulate cavitation for different sets of geometries.This method predicts the cavity interface accurately.

Selection of an appropriate turbulence model is another crucial issue for accurate simulation of the cavitation because the cavitation is an unsteady phenomenon usually occurring in high Reynolds number flows. Two turbulence approaches, large eddy simulation (LES) and k-ω SST have been most widely used to simulate cavitating flow. LES regularly allows for medium-scale to small-scale energy transfer that can capture flow mechanisms with much detail for accurate prediction of the cavitation.

Literature survey shows that numerical simulation of cavitation attracted the attention of researchers during the last decades. Kunz et al. (2000) considered cavitation around

* Corresponding author. Tel.: +98 51 38805136; fax: +98 51 38763304.
E-mail address: e.roohi@ferdowsi.um.ac.ir (E. Roohi).

http://dx.doi.org/10.1016/j.oceaneng.2015.12.010

Nomenclature

B	Unresolved transport term in LES
C_ε,C_K	LES empirical constant coefficients
C_e,C_c	Schnerr–Sauer mass transfer model constants
$CD_{k\omega}$	Positive portion of the cross-diffusion
C_{dest}, C_{prod}	Kunz mass transfer model constants
C_{d0}	Constant in the drag coefficient for a conical cavitator
C_x, C_{x0}	Functions (given by Eqs. (36) and (37))
D	Cavity diameter
d	Cavitator diameter
$\boldsymbol{D}$	Rate of strain tensor
$\tilde{\boldsymbol{D}}_{\boldsymbol{D}}$	Eddy diffusivity tensor
$F1$, $F2$	Turbulence Functions (given by Eqs. (14) and (18))
G	Filter function
H	Height of the cone cavitator
$\boldsymbol{I}$	Unit tensor
k	Kinetic energy
L	Cavity length
l	Liquid
$\dot{m}$	Mass transfer rate between the phases
n_0	Initial number of bubbles
P	Pressure
R	radius of Cylinder
Re	Reynolds number
R_b	Radius of bubbles
s	Viscous stress tensor
$t\infty$	Mean flow time
U	Velocity magnitude
νSGS	Subgrid scale viscosity
X_j	Components of the Cartesian coordinate
y	Distance the between surface
σ	Cavitation number
∞	Free stream value
$\alpha\pi$	cone half-angle
ϕ	volumetric flux
ρ	Density
ϑ	Vapor
δ	Re-entrant jet length
v	Velocity vector
μ	Viscosity
Δ	Filter width
γ	Volume fraction
ω	Vorticity
μ_k	Viscosity of the vortex
β^*,$\sigma_{\omega 2}$,α, β, c_μ	Constant coefficients for the k-ω sst turbulence model
–	Averaging

submerged objects for the axisymmetric and steady state condition with multiphase computational fluid dynamics. They conveyed different parameter such as pressure distribution, drag coefficient and cavity shape and compared with the experimental data. Baradaran Fard and Nikseresht, (2012) simulated unsteady 3-D cavitating flows around a cone and disk cavitator. RANS equations and an additional transport equation for the liquid volume fraction are solved using a finite volume approach through the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm. For the implementation of the turbulent flow, k-ω SST model was used. The results are in good agreement with experimental data and analytical relations. Guo et al. (2011) simulated the cavitating flow around an underwater projectile with natural and ventilated cavitation based on the homogeneous equilibrium flow model, a mixture model for transport equation and a local linear low-Reynolds-number k-ε turbulence model. Shang (2013) simulated cavitation around the cylindrical submarine. They used K-ω SST for turbulence model, VOF method for the cavity interface reconstruction and the Sauer model for mass transfer to capture the cavitation mechanisms within broad ranges of cavitation numbers from 0.2 to 1.0. Park and Hyung (2012) simulated high-speed supercavitating flows around a 2-D symmetric wedge-shaped cavitator and hemispherical head form body using an unsteady Reynolds-averaged Navier–Stokes equations solver based on a cell-centered finite volume method. The computed result compared with an analytical solution and numerical results using a potential flow solver. Yu et al. (2014) simulated dynamic behaviors of cavitation over a 3-D projectile at the cavitation number σ=0.58 based on LES, k−μ transport equation and VOF method with the Kunz model for the mass transfer. Evolution of cavitation in simulation was consistent with the experimental. Chen et al. (2015) investigated the collapse regimes of the cavitation on the submerged vehicles navigating with continuous deceleration in the range of $0.2 \le \sigma \le 0.5$. A homogeneous equilibrium cavitation model that combined with the pressure–velocity–density coupling algorithm was used to simulate the cavitating flows. There are some recent works witch report cavitation phenomena using advanced turbulence models. For example, Decaix and Goncalves (2013) used a compressible, multiphase, one-fluid RANS solver to study turbulent cavitating flows. Ji et al. (2013) simulated cavitating turbulent flow around hydrofoils by using the Partially-Averaged Navier–Stokes (PANS) method and a suitable mass transfer cavitation model. Their predicted cavity characteristic compared well with experimental data. Zhang and Khoo (2014) developed a pressure-based compressible-medium numerical method to perform computations of the cavitating flow. They demonstrated that their method is capable of simulating the dynamics of unsteady cavitating flow. Ji et al. (2014) investigated numerically the structure of the cavitating flow around a twisted hydrofoil using a mass transfer cavitation model and a modified RNG k-ε model with a local density correction for turbulent eddy viscosity. Cavity structures and the shedding frequency agreed fairly well with experimental observations. Goncalves and Charriere (2014) proposed an original formulation for the mass transfer between phases to study one-dimensional inviscid cavitating tube problems. Numerical results are given for various inviscid cases and unsteady sheet cavitation developing along venturi geometries and compared with experimental data. Ji et al. (2015) studied the behavior of cavities around a NACA66 hydrofoil numerically by using Large Eddy Simulation (LES) coupled with a homogeneous cavitation model. Various fundamental mechanisms governing the complex cavitating flow behaviors, including the cavitation shedding dynamic evolution, cavitation–vortex interaction and cavitation excited pressure fluctuation, were examined and summarized.

In this research, we consider cavitation and supercavitation over hemispherical head-form body and a conical cavitator using an open source package, that is, OpenFOAM. We used Kunz and Sauer mass transfer models combined with both of the LES and k-ω SST turbulence models to simulate cavitating flows. A compressive velocity form of the volume of fluid (VOF) method is employed to track the interface of liquid and vapor phases.