International Journal of Multiphase Flow 98 (2018) 1–23

Contents lists available at ScienceDirect

International Journal of Multiphase Flow

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

Cavitation characteristics around a sphere: An LES investigation

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 16 January 2017
Revised 11 August 2017
Accepted 26 August 2017
Available online 30 August 2017

Keywords:
Unsteady cavitation
Sphere
Large eddy simulation (LES)
Wake
Vortical structure

ABSTRACT

Here we examine partial and supercavitation over a sphere at a constant Reynolds number of 1.5×10^6 and a broad range of cavitation numbers ($0.36 < \sigma < 1$). Large eddy simulation (LES) and Sauer mass transfer model were used to simulate the dynamic and unsteady cavitation around the sphere. Also, the compressive volume of fluid (VOF) method is used to track the cavity interface. The two-phase flow solver of the OpenFOAM package, *intephaseChangeFoam* is employed. Large-eddy simulation of cavitating flow over the sphere is compared with the non-cavitating flow at the same Reynolds number. This work provides a thorough understanding of the fluid dynamic characteristics of the sphere cavitation such as vorticity field, turbulent kinetic energy, pressure, velocity, streamlines and boundary layer. Also, detailed analyses of the instantaneous cavity leading edge and separation point location, vortex shedding, streamwise velocity fluctuation and evolution of the cavity are reported. Characteristics of the wake of the cavitating flows are compared with the single-phase results. We report that cavitation suppresses instability in the near wake region and delays the three-dimensional breakdown of the vortices. The volume fraction contours of the cavity cloud obtained from the numerical simulations are compared with the experimental data at the same working condition with a suitable quantitative accuracy.

1. Introduction

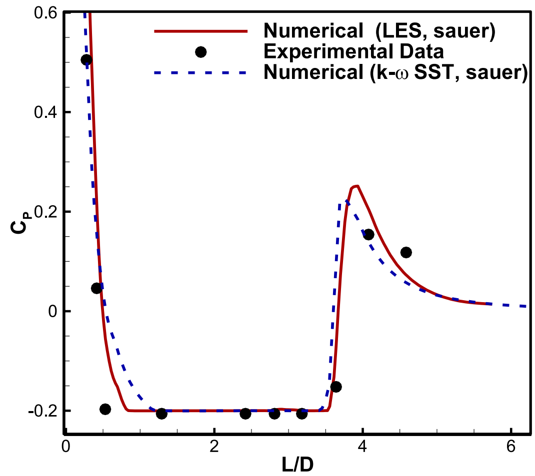
Cavitation is attributed to the formation of vapor when the local liquid pressure becomes lower than its saturated vapor pressure. Cavitation is considered as an unsteady, three-dimensional, discontinuous or periodic, multiphase and complex physical phenomenon. Cavitation often happens in hydraulic devices such as turbines, pumps, pipe systems, fuel injectors, underwater vehicles, submarine, hydrofoils and marine propeller blades as a basic property of the liquid. A dimensionless number, i.e., cavitation number, $\sigma = (P_\infty - P_v)/0.5\rho U_\infty^2$ categories cavitation, where P_v is the vapor pressure, ρ is the liquid density, and P_∞, U_∞ are the free stream flow pressure and velocity, respectively. Depending on the value of the cavitation number, several cavitation regimes were reported in liquid flows, i.e.: incipient cavitation, shear cavitation, sheet/cloud cavitation, and supercavitation.

In the cloud cavitation, the formation, detachment, and collapse of unsteady or periodic sheet cavities occur around the cavitating body. Cloud cavitation with its unsteady nature has considerable consequences on hydraulic and marine equipment, including unsteady behaviors, noise, vibration, and erosion. Supercavitation occurs due to increased velocity of the moving body and consists of a long and steady cavity region. It can decrease the drag of an underwater high-speed moving body, thus enabling it to move with a quite higher speed under the water. In general, cloud and supercavitation attracted the attention of several researchers during the past and recent decades.

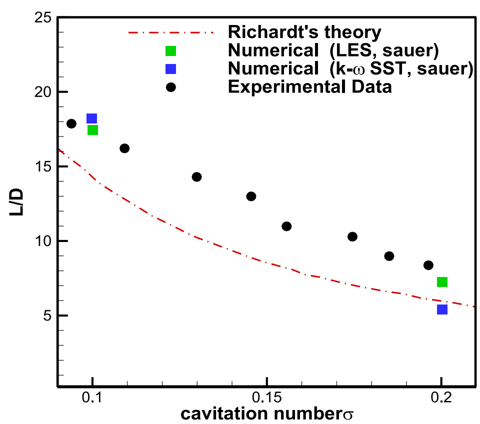
Numerical and experimental investigations of cavitating flow are of great interest to researchers during the past few years. For example, Franc and Michel (2004) studied the role of the unsteady cavity closure and re-entrant jet formation in the cloud cavitation. Brandner et al., (2010) investigated cloud cavitation around a sphere experimentally at $Re = 1.5 \times 10^6$ with cavitation numbers varying between 0.36 and 1.0. They investigated the instantaneous location of the cavity leading edge, separated laminar boundary layer, shedding mechanism, and shedding frequency. Shang et al. (2012) performed numerical simulations of cavitating flow over a sphere using the Large Eddy Simulation (LES) approach together with a mixture assumption and a finite rate mass transfer model. However, they just compared a 3D view of the volume fractions contours of the vapor phase with the experimental data (Brandner et al., 2007) at the same working condition. Roohi et al. (2013) simulated supercavitating flows over a hydrofoil using the LES approach and the Volume of Fluids (VOF) technique. Ji et al. (2013) simulated cavitating turbulent flow around hydrofoils using the Partially-Averaged Navier–Stokes (PANS) method and a suitable mass transfer cav-

* Corresponding author.
E-mail address: e.roohi@ferdowsi.um.ac.ir (E. Roohi).

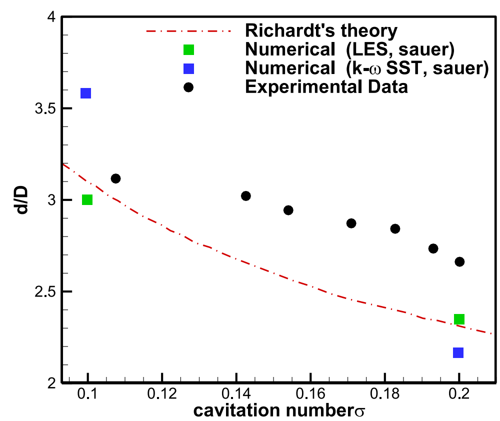
http://dx.doi.org/10.1016/j.ijmultiphaseflow.2017.08.013
0301-9322/



(a) C_p distribution over the hemispherical head-form body at σ=0.2.



(b) Normalized cavity length behind the disk.



(c) Normalized cavity diameter behind the disk.

Fig. 1. The i*nterPhaseChangeFoam* validation for cavitating flows over different geometries.

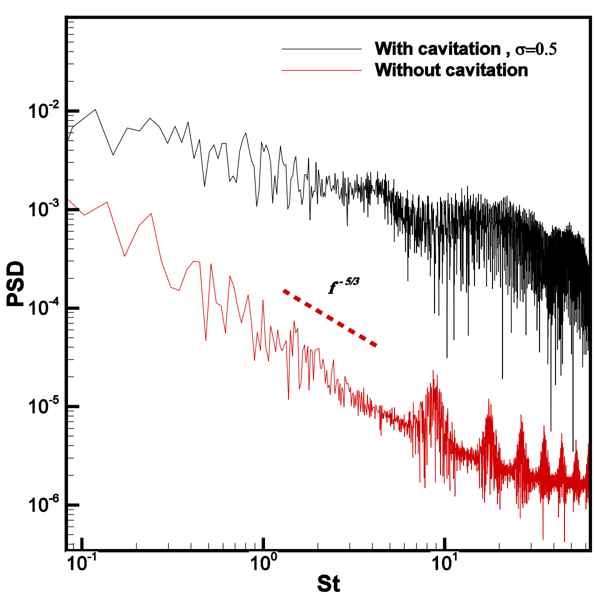


Fig. 2. The Power spectrum density (PSD) analysis for the drag coefficient over the sphere for cavitating at σ =0.5 and non-cavitating flow.

itation model. The predicted cavity characteristic compared well with the experimental data. Shang (2013) simulated cavitation around the cylindrical submarine. He used K-ω SST turbulence model with VOF method and Sauer mass transfer model to capture the details of the cavitation mechanisms within broad ranges of cavitation numbers. Ji et al. (2014) numerically investigated 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. Yu et al. (2014) simulated the dynamic evolution of cavitation over 3-D geometries using the LES and k−ε turbulence approaches as well as VOF technique with the Kunz mass transfer model. Chen et al. (2015) considered the collapse regimes of the cavitation on the submerged vehicles with a constant deceleration. Roohi et al. (2016) considered cavitating and supercavitating flow behind a 3D disk with specific emphasis on detailed comparisons of the various turbulence and mass transfer models. Cheng et al. (2016) investigated the unsteady cavitating turbulent flow around twisted hydrofoil using Zwart cavitation model combined with a filter-based density correction model (FBDCM). Their numerical results simulated the entire process of cavitation shedding including the re-entrant jet accurately in comparison with the experimental data. Gnanaskandan and Mahesh (2016a,b) studied sheet to cloud cavitation transition over a wedge at Re =200,000 and σ =2.1. The frequency of the shedding process, mean pressure and velocity fluctuations were reported accurately. Luo et al. (2016) summarized the recent progress for the cavitation study in the hydraulic machinery including turbo pumps and hydro turbines. Wang et al. (2016) studied unsteady cloud cavitation around