Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
2017,29(6):1089-1092
DOI: 10.1016/S1001-6058(16)60823-5
www.sciencedirect.com/
science/journal/l0016058

Compressible effect on the cavitating flow: A numeric study *

Wei Zhang (张伟) [1], Xiao-dong Bai (柏晓东) [2], Zheng Ma (马峥) [3], Gang Chen (陈刚) [1], Yong Wang [4]
1. *Science and Technology on Water Jet Propulsion Laboratory, Marine Design and Research Institute of China, Shanghai 200011, China, E-mail: waynezw0618@163.com*
2. *Ministry of Education Key Laboratory of Structural Dynamic Behavior and Control, School of Civil Engineering, Harbin Institute of Technology, Harbin 150090, China*
3. *China Ship Scientific Research Center, Wuxi 214082, China*
4. *Max Planck Institute for Dynamics and Self-Organization, Am Faßberg 17, Göttingen 37077, Germany*

(Received September 10, 2017, Revised October 17, 2017)

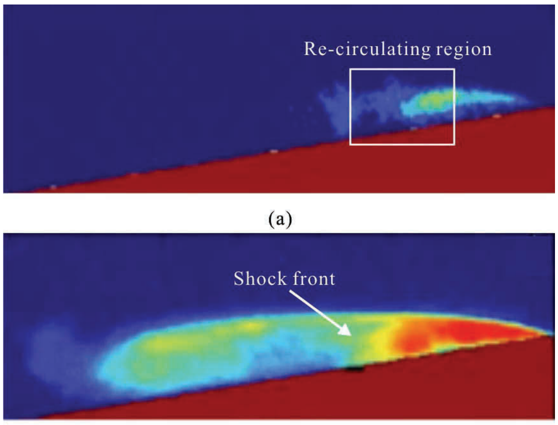
Abstract: To understand the effect of the compressibility on the cavitating flow, a compressible, multiphase, single component Reynolds averaged Navier-Stokes (RANS) solver is used to study the cavitating flow on a wedge in the present work. A barotropic equation of status is used. A non-linear model for compressibility in the mixture is adopted to capture the effect of the compressibility within the complex cavitation bubbly mixtures. An unsteady cavitation phenomenon is found in the numerical simulation. The numerical results of local compressibility and Mach number in the bubbly mixture are given. The mechanism responsible for the unsteady shedding of the bubbly mixture is discussed based on the numerical results.

Key words: Cavitation, compressibility, barotropic flow

Cavitation is an important phenomenon in industry. In most cases, marine propeller, pump, diesel injector etc. are suffered from cavitation because of the cavitation-induced vibrations, noises, erosions and so forth[1].

Phase change from liquid to vapor in the low-pressure region is the main feature of the cavitation. If the low-pressure region is located at downstream of a bluff body, the cavitation produced by separation could be unstable. Such phenomenon is believed to be characterized by the re-entrant jets[2,3], as shown in Fig.1(a). Many experimental measurements have been carried out to study the re-entrant jets and the cavitation[4-6]. Most recently, Ganesh et al.[7] found experimentally bubbly shock propagation, as shown in Fig.1(b), indicating that compressibility of the bubbly mixture could be another key factor of unstable cavitation.

Although there are many numerical simulations carried out to study the cavitation, only a few considered the compressibility. Goncalvès developed a one-fluid compressible Reynolds averaged Navier-Stokes (RANS) solver with a simple equation of state (EOS) based on a sinusoidal barotropic law for the mixture[8]. He used the solver to compute a periodic unsteady cavitation on a wedge[9], and only re-entrant jet flow was captured. To the best of our knowledge, there is no published numerical work concerning the com-



(a)

(b)

Fig.1 (Color online) Two different types of unstable cavitation on the wedge, from the supplemental material of Ref.[5]

---

* **Biography:** Wei-Zhang (1981-), Male, Ph. D., Engineer
Corresponding author: Gang Chen,
E-mail: gangchen@sjtu.edu.cn

pressibility of the mixture in the cavitating flow so far.

To understand the compressible effect on the cavitating flow, the compressible, multiphase, single component RANS solver, cavitatingFoam, developed by Karrholm et al.[10] within the open source CFD package, OpenFOAM, is used in the present work. To close the mathematic model, a barotropic EOS is chosen to link the pressure with the density. The barotropic EOSs for pure liquid and pure vapor are given below:

$$\rho_v = \psi_v p \tag{1}$$

$$\rho_l = \rho_{l,0} + \psi_l p \tag{2}$$

where the subscripts v and l refer to vapor and liquid respectively. ψ refers to the compressibility, and is the inverse square of the speed of sound C. The mixture of bubbly flow is assumed to be in homogenous equilibrium. The local fraction of the vapor, γ, is computed as below

$$\gamma = \frac{\rho - \rho_{l,\mathrm{sat}}}{\rho_{v,\mathrm{sat}} - \rho_{l,\mathrm{sat}}} \tag{3}$$

Karrholm et al.[10] used a linear interpolation function to calculate ψ for mixture based on the fraction of the vapor, γ. However, because of the complex nonlinear effect of the compressibility in the mixture, such linear interpolation may not be good enough. For instance, the speed of sound for water and water vapor are 1 480 m/s and 348 m/s respectively at 20°C, while that of the mixture can be as slow as 20 m/s. In other words, linear interpolation function does not represent the underlying physics and cannot be used to simulate the compressible effects of the bubbly mixture especially for cavitating flow under the influence of the compressibility. Brennen introduced a model by assuming the new equilibrium could be rapidly established when a small pressure change occurred in the mixture[11]. And according to his work, the compressibility ψ can be calculated based on the local volume fraction of vapor γ as below

$$\psi(\gamma) = [\gamma\rho_v + (1-\gamma)\rho_v]\left(\frac{\gamma}{\rho_v\psi_v} + \frac{1-\gamma}{\rho_l\psi_l}\right) \tag{4}$$

Based on this equation, ψ can be plotted as a function of γ for diesel fuel. As shown in Fig.2, the local speed of sound could be slower than 20 m/s in the mixture. As a result, high local Mach number could be expected in the cavitating flow of diesel fuel even when the character speed is not too large. In fact, Ganesh et al.[7] used a modified version of this formula to estimate the local speed of sound in their measurement, and then got shock propagation as the conclusion.

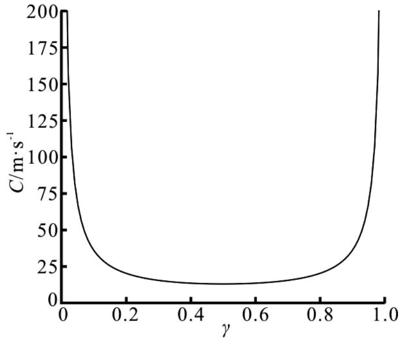


Fig.2 Speed of sound as a function of vapor fraction

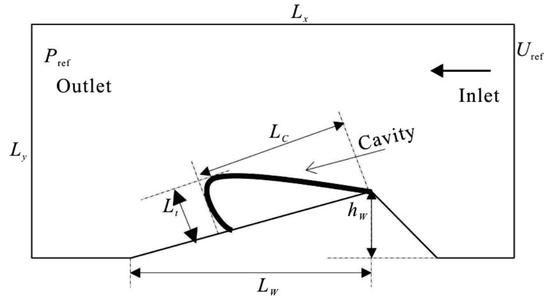


Fig.3 The schematic view of the domain

The Brennen's model for speed of sound in the mixture is adopted here to simulate the cavitating flow on a wedge. Ganesh et al. performed experimental measurements for this case. The schematic view of the domain for the present simulation is given in Fig.3. A quasi-2D domain is considered in the present work. The length and the height of the wedge, L_W and h_w, are 180 mm and 25.4 mm respectively. The domain size $L_x \times L_y$ is $60L_W \times 28L_W$. L_t and L_c are the maximum cavity thickness and the maximum cavity length, respectively. A mesh with 151 657 hexahedra cells is used here. The mesh is refined around the wedge region with refineMesh in OpenFOAM. Just one layer is considered in the 3rd direction, and the front and back directions are set to be empty boundary condition. A fixed value U_{ref} is given for velocity at the inlet of the domain. The pressure at the outlet boundary is set to be the reference pressure P_{ref}. Diesel fuel is considered in our simulation. The Reynolds number, $Re = U_{\mathrm{ref}} L_W / \nu$, is set to be 1.44×10^6, where ν is the kinematic viscosity of fuel. The cavitation number, $\sigma = (P_{\mathrm{ref}} - P_{\mathrm{sat}})/(0.5\rho U^2)$, is set to be 1.95, where P_{sat} is the vapor pressure of diesel fuel. These two parameters are the same as those used in Ganesh et al.'s experiment. The speed of