International Journal of Heat and Mass Transfer 86 (2015) 294–301

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Calculation and verification of dynamical cavitation model for quasi-steady cavitating flow

Zhang XiaoBin, Zhu JiaKai, Qiu LiMin, Zhang XueJun *

Institute of Refrigeration and Cryogenics, Zhejiang University, Hangzhou 310027, PR China

ARTICLE INFO

Article history:
Received 23 March 2012
Received in revised form 16 January 2015
Accepted 21 January 2015
Available online 21 March 2015

Keywords:
Cavitation model
Hydrofoil
Bubble radius
CFD

ABSTRACT

Due to the intrinsically unsteady characteristics of bubbles, the spontaneous cavitation flow essentially is an unbalanced hydrodynamic process. The so called full cavitation model (FCM) is now widely used in the modeling of cavitating flow, as it accounts for all the first-order effects, but assumes the typical bubble radius is the same as the maximum possible size based on the flow balance condition between aerodynamic drag and surface tension forces. The present paper develops a new cavitation model (named the dynamic cavitation model, DCM) by combining an expression for the pressure-dependent bubble radius with the FCM. The expression was derived from the Gibbs–Duhem equation and Young–Laplace equation under the assumption of thermodynamic equilibrium. Subsequently, the DCM was implemented into a homogeneous mixture flow model-based CFD code for cavitation flow. Verification calculations for water cavitating flows over a hydrofoil and submerged cylindrical bodies with different forehead geometries were performed to validate the DCM by comparing it with the experimental results. The quasi-steady pressure distributions computed with DCM accorded well with the results of FCM and the experiments. However, because the DCM is more sensitive to pressure than the FCM, the cavitation zone computed using the DCM spreads out over a smaller volume than that computed from the FCM.

© 2015 Published by Elsevier Ltd.

1. Introduction

Cavitation is defined as the formation of vapor bubbles in a flow when the local pressure falls below the liquid's saturation vapor pressure. It is a common phenomenon in fluidic machineries, such as inducers, turbopumps, hydrofoils, etc. In most cases, cavitation is undesirable, and can lead to damage, performance degradation, and even system failure [1].

Computational methods for spontaneous cavitation have been studied for over several decades [2–4], and can be generally divided into two categories: single-phase modeling with cavitation interface tracking, and multi-phase modeling with an embedded cavitation interface. In the former method, only the liquid phase is modeled with the assumption that the vapor pressure in the cavity region is constant, corresponding to the local temperature. Although, it requires a considerable amount of preliminary knowledge, many successful applications have been reported [5,6]. The latter method models both phases via one of two approaches: a two-fluid model, which considers separate conservation equations for each phase [7], or a mixture model, which regards the two-phase mixture as a single-fluid [8]. In the mixture model, since cavitation primarily occurs in the low-pressure regions, where the velocity is relatively high, the slip velocity between the phases is commonly not considered in most numerical simulations, resulting in the so called the homogeneous mixture condition [9]. In contrast, the nonhomogeneous mixture model (also called the drift-flow model) considers the slip velocity. Examples of this approach are provided in the works by Rhee et al. [10], which uses an algebraic relation to calculate the slip velocity in the modeling of propeller cavitation. It is worth mentioning that all of the above models for cavitation calculations are based on the pressure equilibrium condition between the phases, which is a proven assumption in modeling the spontaneous cavitating flow in the fluidic machineries. However, solutions based on a non-equilibrium pressure distribution, such as when cavitation results from a shock, were also reported [11,12].

The definition of the variable density field in the cavitation process provides the primary difference between the various approaches for the homogenous mixture model. One approach uses a barotropic equation of state, together with the equation of the speed of sound to couple the pressure and density [13,14]. Recently, a gas volume/mass fraction transport equation-based method, based on the homogenous mixture model [9,15,16] has

* Corresponding author. Tel./fax: +86 571 87952446.
E-mail address: xuejzhang@zju.edu.cn (X. Zhang).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.01.098
0017-9310/© 2015 Published by Elsevier Ltd.

become popular. In this method, the information regarding the vapor volume/mass fraction distribution is modeled by a transport conservation equation. One apparent advantage of this model comes from the potential for modeling the impact of inertial forces on cavities like elongation and drift of cavity bubbles, such as is demonstrated in the works by Singhal et al. [17], Merkle et al. [18], and Kunz et al. [19]. These works, that can be called cavitation models, differ in their calculation of the source terms in the vapor fraction transport equation. Table 1 lists the mathematical expressions of the cavitation models in a chronological order of publication [17–20]. The first two cavitation models are largely dependent on empirical judgment, inducing much uncertainty for cavitation in various fluids [8]. The IDCM model assumes that there is a clear interface between vapor and liquid in the cavitation zone, a feature that is not suitable for calculating cavitation in cryogenic fluids, that display close characteristics between the liquid and vapor phases [21]. The so called full cavitation model is capable of modeling cavitation for nonthermosensitive [17] and thermosensitive fluids [15,22], and has been adopted by the commercial CFD software code Fluent. Here, the single bubble radius in the model is set as the maximum possible bubble dimension assuming the size is determined from a balance between aerodynamic drag and surface tension forces. Note that the model does not include the effects of the pressure on the bubble radius. In essence, the bubble is intrinsic unsteady and periodically appears, grows, coalesces and finally collapses in the cavitation closure zone. Therefore, both the rate of change and the bubble size itself are highly pressure-dependent.

The aims of the present paper are to develop a new cavitation model, which considers the effects of the pressure on both the rate of change and the size of the bubble. The pressure-dependent bubble radius is obtained by combining the Gibbs–Duhem equation and Young–Laplace equation with the assumption of thermodynamic equilibrium during the cavitation process. Then, it is used to re-calculate the radius in the FCM, thereby obtaining the dynamic cavitation model (DCM). The CFD framework for cavitation is built and solved using Fluent, based on the homogenous mixture model and vapor fraction transport-based equation. The DCM is implemented into the iterations by the method of User-defined Functions. Steady cavitating flow over several geometries, including the NACA66 (MOD) hydrofoil, and submerged cylindrical bodies with different forehead geometries are modeled with the DCM and FCM, respectively. Comparisons of the pressure distributions with the published experimental data validated the DCM.

2. Mathematic framework

For water cavitation, since the ratio of liquid to vapor density is over 40000 at a temperature of 20 °C, small amounts of vaporized liquid can make up a large pressure drop, primarily from the local exchange between the static and the dynamic pressure. In addition, the specific heat for a unit volume of water is large enough to make the temperature change negligible. Therefore, the energy equation is usually not solved, which greatly decreases the convergence difficulties for numerical iterations. The set of governing equations for cavitation under the homogenous mixture model includes the conservative form of the Navier–Stokes equations, the κ–ε two-equation turbulence closure, and a transport equation for the vapor mass fraction. The continuity and momentum equations for steady state are given below [8]:

$$\frac{\partial(\rho_m)}{\partial t}+\frac{\partial(\rho_m u_j)}{\partial x_j}=0 \tag{1}$$

$$\frac{\partial(\rho_m u_i)}{\partial t}+\frac{\partial(\rho_m u_i u_j)}{\partial x_j}=-\frac{\partial P}{\partial x_i}+(\mu+\mu_t)\times\frac{\partial}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right) \tag{2}$$

Where x and the subscripts i, j and k denote the coordinate axes, t is time, ρ_m is the mixture density, defined as $\rho_m = \rho_v\alpha_v + \rho_g\alpha_g + \rho_l(1\text{-}\alpha_v\text{-}\alpha_g)$, (subsequently, the subscript m for all mixture variants will be omitted for brevity), α represents the volume fraction, and the subscripts v, g and l denote, respectively, the vapor phase, non-condensable gas and liquid phase, u represents the velocity vector, P is pressure, μ is viscosity, and the subscript t denotes turbulent flow. The effect of slip velocity between liquid phase and vapor phase on the momentum exchange has been neglected, because cavitation often occurs in the high-speed flow region.

The κ–ε two-equation turbulence model has been widely used for simulating the quasi-steady sheet cavitating flow for nonthermosensible and thermosensible fluids [8,22]. Compared with the standard κ–ε model, the realizable κ–ε turbulence model has shown substantial improvements in its ability to characterize flows with sharp streamline curvature or vortices. In this paper, the realizable κ–ε model with enhanced wall treatment is used to investigate turbulent mixing. The sensitivity of the turbulent computations to the wall grid resolution, via wall treatments was analyzed in our previous paper [22]. The wall treatment can provide a better pressure distribution than the standard wall function, because it assumes that a local equilibrium exists between the production of kinetic energy and its dissipation rate at the wall-adjacent cells [22].

3. Dynamic cavitation model

The distribution of the vapor mass fraction, f, in the cavitation process is determined by solving the transport equation as follows:

$$\frac{\partial}{\partial t}(f\rho)+\nabla\cdot(f\rho u_g)=\dot{R} \tag{3}$$

Where $\dot{R}$ is the net evaporation rate or condensation rate that depends on the local conditions. Singhal et al. [17] have used the

Table 1
Cavitation models for CFD simulations.

Model and time reported	Vapor evaporation item R^+	Vapor condensation item R^-
Merkle et al. (1998) [17]	$\frac{C_{prod}MAX(p-p_v,0)(1-\alpha_l)}{(0.5\rho_l U_\infty^2)t_\infty}$, $C_{prod}=80$	$\frac{C_{dest}\rho_l MIN(p-p_v,0)\alpha_l}{(0.5\rho_l U_\infty^2)t_\infty}$, $C_{dest}=1$
Kunz et al. (2000) [18]	$\frac{C_{prod}\alpha_l^2(1-\alpha_l)}{\rho_l t_\infty}$, $C_{prod}=3\times10^4$	$\frac{C_{dest}\rho_v MIN(p-p_v,0)\alpha_l}{(0.5\rho_l U_\infty^2)\rho_l t_\infty}$, $C_{dest}=1$
IDCM (2004) [19]	$\frac{C_{prod}MAX(p-p_v,0)(1-\alpha_l)}{(0.5\rho_l U_\infty^2)t_\infty}$ $\frac{C_{prod}}{0.5\rho_l U_\infty^2}=\frac{1}{(\rho_l-\rho_v)(U_{v,n}-U_{l,n})^2}$	$\frac{C_{dest}\rho_l MIN(p-p_v,0)\alpha_l}{(0.5\rho_l U_\infty^2)\rho_v t_\infty}$ $\frac{C_{dest}}{0.5\rho_l U_\infty^2}=\frac{1}{(\rho-\rho_v)(U_{v,n}-U_{l,n})^2}$
Full cavitation model (2005) [16]	$\frac{C_{prod}\sqrt{k}}{\sigma}\rho_l\rho_v\left[\frac{2}{3}\frac{MAX(P-P_v,0)}{\rho_l}\right]^{1/2}$, $C_{prod}=0.02$	$\frac{C_{dest}\sqrt{k}}{\sigma}\rho_l\rho_l\left[\frac{2}{3}\frac{MAX(P-P_v,0)}{\rho_l}\right]^{1/2}$, $C_{dest}=0.01$

Download English Version:

https://daneshyari.com/en/article/656739

Download Persian Version:

https://daneshyari.com/article/656739

Daneshyari.com