Computers and Fluids 162 (2018) 1–10

Contents lists available at ScienceDirect

Computers and Fluids

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

An Eulerian stochastic field cavitation model coupled to a pressure based solver

Boxiong Chen*, Michael Oevermann

Division of Combustion, Chalmers University of Technology, Gothenburg, Sweden

ARTICLE INFO

Article history:
Received 18 July 2017
Revised 10 October 2017
Accepted 4 December 2017
Available online 6 December 2017

Keywords:
Multiphase flow
Cavitation
Volume of fluid
Eulerian stochastic field method

ABSTRACT

Probability density functions (PDF) and the relevant methods have been widely used to describe non-linear phenomena in the realm of turbulence modelling and CFD. In order to solve PDF transport equations, the main trend of previous studies rely on Monte Carlo method with Lagrangian particle tracking. However, as with any Lagrangian based approach, the scalability of the parallelized simulations of such method is less than satisfactory. An Eulerian stochastic field model has been presented recently by Dumond et al. [1] to simulate cavitating flows. Their model uses a fully compressible density based solver. Here we present an adapted version using an iso-thermal cavitation model adopting the homogeneous mixture assumption in a pressure based flow solver which is more relevant to engine simulations. A PDF method is used to represent a distribution of vapour volume fractions, based on which the Eulerian stochastic field (ESF) method is applied to perform a three-dimensional large eddy simulation (LES) of the cavitation phenomena inside an academic fuel injector configuration. The numerical model is based on a volume of fluids approach and coupled with a pressure based solver for the flow field, and is implemented in the framework of the open source C++ toolbox OpenFOAM. The result of the ESF simulation is compared against that from a typical single volume fraction solver for validation. Vortex structures and its correspondence to cavitation are shown, and the behaviour of the PDF at different probe locations at different times are acquired to demonstrate the potential of the ESF model in capturing both transient and stochastically steady cavitation.

1. Introduction

Cavitation is a vapourization phenomenon of a liquid following a strong enough local pressure drops, starting at favorable spots in the fluid that are known as nuclei. Nuclei become larger vapour bubbles as they are transported to locations with local pressure below saturation pressure, and collapse where the local pressure is higher than saturation pressure. Cavitation can occur in various engineering applications, and influence the devices and their performances through the pressure wave it induced or its interaction with fluid flow. E.g., in hydraulic machinery cavitation induces corrosion that reduc'es the efficiency and lifespan of the machine. In the realm of internal combustion engines, cavitation happens inside the high pressure injectors nowadays, enhancing the breakup of the fuel jet, which is key to fuel-air mixing and pollutant formation. Since there is an ongoing trend towards higher and higher injection pressures which increases the likelihood of cavitation inside the nozzle, studies on cavitation in injector nozzles have been carried out extensively through both experimental and simulation efforts. Sou et al. [2] have demonstrated that the cavitation inside the injection needle enhances spray atomisation, which in turn affects the quality of the subsequent combustion. Experimental studies of cavitation on realistic nozzle geometries and injection conditions are extremely difficult. The geometric size and the residence time of the bubbles require restrictively high resolutions of both time and space and optical access for laser based measurement techniques. In addition, the high injection pressures involved in real injectors are usually hard to resemble on transparent nozzles that are usually made of acrylic materials or quartz glass. Therefore, a number of experimental studies are set up on up-scaled nozzles and low pressure conditions, e.g. [3,4]. Mauger et al. in [5] performed a study on low-pressure, quasi 2D geometry using Shadowgraph-like imaging, Schlieren technique, and interferometry. In [6], Gavaises and Addriotis performed a study on string cavitation with a transparent replica of a five-hole injection nozzle under low pressure conditions. Two High-Speed Digital Video (HSDV) cameras are synchronized to capture the cavitation structures from two views. Images confirmed that string cavitation is a source of variation of spray cone and deflection angles.

* Corresponding author.
E-mail address: chenb@chalmers.se (B. Chen).

https://doi.org/10.1016/j.compfluid.2017.12.002
0045-7930/

Computational studies with various modelling methodologies have been carried out for cavitating flows. A Lagrangian description of individual bubbles or bubble clusters can be used to represent cavitation [7,8]. The interaction between bubbles, and bubble-wall interactions can be modelled up to a certain level of complexity and computational cost. Sou et al. [2] and Giannadakis et al. [8] followed the Lagrangian approach, in which vapour cavities are treated as Lagrangian particles under the assumption of a dilute vapour fraction. In [8], a number of submodels on behalf of many primary physical processes are implemented, among which there is a stochastic model that describes coalescence. It is further demonstrated through numerical experiments that break up phenomena mainly take place near the nozzle inlet, whilst coalescence may take place along the entire liquid passage length. A comprehensive review on a variety of primary physical processes and their modelling relevant to the Lagrangian framework has also been given. In the realm of Eulerian modelling, a major difference among the existing cavitation models stems from the different treatment of the gas–liquid interface. Dabiri et al. in [9,10] applied a level set method in their series of work to resolve the phase boundary between liquid and vapour phase. The onset of cavitation is identified when the maximum tensile stress exceeds a critical value. Based on this assumption a model of the sub-grid shape of the interface is applied. The curvature of the interface is then used to calculate surface tension forces in the momentum equation. Marcer and LeGouez [11] made use of a volume of fluids (VOF) method, reconstructed the interface with planes of arbitrary orientations in each cell. In the above mentioned studies, the sub-grid shape of interface is considered either by direct reconstruction or modelling. Methods of this kind fit well into fundamental studies of the behaviour of only a few bubbles. A more macroscopic class of models, called interface diffuse models [12], regard bubbly flow as a continuous dispersed phase. This approach spends no effort tracking the fronts of each individual bubbles, but regard the bubbly flow as a mixture instead. The phases are represented by volume fractions, and the interface is treated as a zone where the two phases coexists. Since physically bubbles exist on the sub-grid level, the volume fraction field does not define a sharp interface between the phases but provides the volumetric fraction of liquid and vapour within a computational cell. In the category of interface diffuse models, the number of equations that are used to describe the two phase further subcategorize different models. Baer and Nunziato in [13] proposed a seven equation model that consists of a conservative set of equations for mass, momentum, and energy for each of the two phases and an additional transport of equation for the volume fraction. A less complex model by Ishii [14], Boure and Delhaye [15], and Butler et al. [16] applies a six-equation model, which consists of the conservation equations for both phases, but only one pressure is applied assuming either incompressibility of one of the phases, or pressure equilibrium between the two phases. This approach reduced the computational cost involved in solving a volume fraction transport equation at the expense of a reduced validity on problems where transient wave propagation is important. Based on the seven-equation models, Saurel et al. [17] derived a five-equation model, assuming a mechanical equilibrium between the two phases. It involves transport of volume fraction, one set of momentum and energy equation, but two phase balance equations. Rodio and Abgrall [12] applied a discrete equation method (DEM) and a splitting method on the five-equation model to preserve the positivity of the solution and reduce the computational cost at the same time. The even simpler models assume pressure and temperature equilibrium of the phases, thereby regarding the multiple phases practically as one. Yuan et al. in [18,19] and Chen et al. in [20] adopted the volume of fluids method, combined with various turbulence models to describe the flow field. A simplified Rayleigh–Plesset relation [19,21], which reflects spherical bubble dynamics, was used to model the phase change. Along the same lines, Altimira and Fuchs [22] investigated the interaction between cavitation and turbulence. As it is shown in [19], the volume fraction field is associated with only one bubble radius (for each computational cell). It is, therefore, natural to think of a representation, through which a distribution of radii can be rendered. Therefore, models that use probability density functions (PDF) have attained a certain level of popularity. The majority of the previous PDF works related to turbulent flows and CFD simulation were devised to perform simulations of combustion applications. Pope in [23] solved the PDF transport equation by tracking Lagrangian particles in combination with Monte-Carlo method. The spatial position of each particle is included as one of the stochastic variables and evolves according to PDF equation. This approach, therefore, entails no Eulerian grids. In [24], Pope developed a more Eulerian-based approach. The particles reside on a Eulerian field, and move from one node to another following rules that are based on transportation. Although the model is based on Eulerian fields, the transport equation of the PDF is not solved with typical finite difference or finite volume schemes, thus making it inconvenient to apply this method to other applications. A full Eulerian PDF solving framework has been developed in [25], and applied successfully on cavitation simulation in [1]. Applying certain stochastic procedures, this approach uses an ensemble of Eulerian fields to represent a distribution of radii.

In the current work, we combine the Eulerian Stochastic Field model for volume fraction with an incompressible solver that is based on pressure-velocity coupling in order to avoid the high computational cost typically involved in a density based solution. A solver is developed on the basis of open source C++ toolbox OpenFOAM extended 3.1. A simulation of the flow through a well-known academic throttle geometry [3,22] is performed, the result of which is then compared against that from the single volume fraction field solver (interPhaseChangeFoam) as a validation. The paper is organized as follow: in the first section, the mathematical model will be presented. This is followed by a detailed description of the stochastic model. The computational setup and the numerical details, especially the stochastic integration schemes, will be discussed thereafter. Next comes the presentation of the numerical simulation on the chosen throttle geometry. Further results and discussions on the bubble size PDF and its transient behaviour are provided. Conclusions and an outlook on the model are then given in the last section.

2. Mathematical and numerical model

2.1. Governing equations for the mixture

In the current work, the two phase flow is regarded as a homogeneous mixture of liquid-vapour, therefore only one set of mass and momentum equation is involved for the mixture:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0, \tag{1}$$

$$\frac{\partial}{\partial t}(\rho \vec{u}) + \nabla \cdot (\rho \vec{u}\vec{u}) = \nabla \cdot (\mu \nabla \vec{u}) - \nabla p, \tag{2}$$

where $\vec{u}$ denotes the velocity, p the static pressure. Here we assume the mixture to be isothermal, therefore there is no energy equation involved in the solving system, and pressure p will be determined by pressure–velocity coupling. The density ρ and dynamic viscosity μ are closed by the following relation following the homogeneous mixture assumption:

$$\rho = (1 - \alpha_l)\rho_v + \alpha_l \rho_l, \tag{3}$$

$$\mu = (1 - \alpha_l)\mu_v + \alpha_l \mu_l, \tag{4}$$