A multi-scale Eulerian-Lagrangian model for simulation of cavitating flows

A multi-scale Eulerian-Lagrangian model for simulation of cavitating flows

Ebrahim Ghahramani and Rickard E. Bensow
Chalmers University of Technology Gothenburg/Sweden
ebrahim.ghahramani@chalmers.se

1 Introduction

Cavitation is a common phenomenon in industrial hydraulic systems, such as marine propulsion systems and fuel injectors. It is in many cases an undesirable and unavoidable occurrence. Cavitation erosion is believed to be the result of violent collapses of the flowing micro-bubbles within very short time scales, which is often accompanied with issues of noise, vibrations, load variations and loss of efficiency in devices such as propellers and pumps. Created by the sheet cavity breakup, bubbly vapour clouds are transported into regions of higher pressure, where collapse-like condensation results in the formation of liquid jets and pressure shocks. Due to the mentioned significance and complexity of the flow field understanding and controlling cavitation has been a major challenge in engineering in recent decades.

In recent years various experimental and numerical methods have been widely used to analyse this complex phenomenon; high-speed video filming has been considered a relatively suitable tool to study cavitation. However, due to the limitations in measuring detailed instantaneous data, cavitation erosion is yet not fully understood using experimental methods. Computational Fluid Dynamics (CFD) simulation, which in the last decade has gained in popularity due to advances in computational resources and modelling, is an alternative to prototype experiments. Numerical simulations can also supplement experimental measurements to have a more comprehensive understanding of the hydrodynamics of cavitation erosion. For example, in a recent study by Gavaises et al. (2015), direct observation of the flow structures was not possible, however vortex shedding was inferred from relevant simulations in the same conditions.Also, Lu et al. (2013) studied propeller cavitation where CFD supported the experimental observations to explain the differences in performance between designs.

Various numerical methods are being used by different researchers today (selectively Bensow and Bark (2010), Giannadakis et al. (2008), Hsiao et al. (2015), Schnerr et al. (2008) and Yakubov et al. (2015)); most of these methods can be categorized in two general approaches. The first approach is based on the mixture equation of state, assuming the thermodynamic equilibrium. In this approach the vapour volume fraction is directly obtained from the cell-averaged fluid state. However, this approach requires very small timesteps since it considers compressible liquid and vapour phases. Therefore, even if there are suitable models that can adequately estimate the behaviour of cavitation structures, their application in industrial problems (especially the large scale marine industry) is limited, as they require considerably higher computational resources.

The second approach is based on a rate equation for vaporization and condensation. Various numerical models are included in this general classification which may be further categorized in different groups. Both Eulerian and Lagrangian viewpoints can be used to track the vapour structures and their interactions with the liquid phase. One of the widely used Eulerian models is when the flow is treated as a single fluid mixture via the interface capturing Volume of Fluid (VOF) and mass transfer between the phases is defined by explicit source terms. The Eulerian methods perform well in regions with moderate flow changes but in zones of strong, vortical flow they cannot capture cavity transport accurately. One of the main reasons is that, typically, these models utilise the asymptotic form of the well-known Rayleigh-Plesset equation of bubble dynamics (Abdel-Maksud et al., 2010). Another limitations of the Eulerian formulations is that the vapour structures smaller than the grid size, e.g. cavitation nuclei and bubbles, cannot be handled exactly.

The Lagrangian models, on the other hand, enable more detailed formulations for transport, dynamics and acoustics of discrete vapour bubbles. These models, based on a more accurate form of the Rayleigh-Plesset equation, are more accurate for cavitating flows with large values of vorticity and pressure gradients. While the bubble sizes in this viewpoint can be much smaller than the grid size, these models are sometimes quite computationally expensive, and cannot represent large non-spherical vapour structures of the size of computational cells or larger.
Considering the abovementioned capabilities and limitations of the Eulerian and Lagrangian formulations, a solution can be to develop a hybrid multi-scale model that is capable in both resolving the large vapour structures and capture the small-scale bubbles. There are a few studies in the literature that follows this method, primarily Vallier (2013) and Hsiao et al. (2015). In the current study a multi-scale model similar to the work of Vallier (2013) is implemented in OpenFOAM. In this model, the large vapour structures are handled using the Eulerian single fluid mixture method and the small scale spherical bubble are tracked in the Lagrangian framework. Also, a criterion for transition between the Eulerian and Lagrangian vapour structures is defined. The new model is developed in the open source C++ package OpenFOAM by improving the InterPhaseChangeFOAM solver.
In the following sections a more detailed expression of the developed model and a qualitative validation of its performance are presented.

2 Method

As mentioned above, a multi-scale model that uses the strength of both the Lagrangian and Eulerian formulations is developed. In this model, for the continuum liquid phase, the continuity and Navier–Stokes equations are solved and the vapour phase can be treated in either a Eulerian or a Lagrangian framework based on the length scale of the structure. One feature of the VOF method is that it treats structures that are smaller than the grid size as a homogenous mixture, thus sparse vapour clouds or subgrid inhomogeneity in cavitation clouds are not well treated. An extremely high mesh resolution is required to capture the small individual cavitation bubbles. Thus, as a solution, we here combine the Eulerian mixture formulation with a Lagrangian model to account for evolution of individual bubbles aiming for a more realistic estimation of the whole range of cavity sizes. Small bubbles can be identified from the mixture solution at each timestep, and transferred to a Lagrangian framework.

In this section the numerical methods in the Eulerian and Lagrangian formulations and the transition criterion are described.

2.1 Eulerian model

In the Eulerian formulation, the VOF method is used to simulate the vapour transport. This method is suitable for modelling large resolvable vapour structures, such as sheet cavitation. A scalar equation is solved for the transport of the vapour volume fraction quantity,

$$\frac{\partial \alpha_1}{\partial t} + u_i \frac{\partial \alpha_1}{\partial x_i} = \frac{\dot{m}}{\rho}, \qquad (1)$$

where, α_1 is the liquid volume fraction. The right hand side source term represents the rate of vaporisation / condensation of water. Various formulations have been suggested for this source term; in this study the Sauer-Schnerr method is used (Schnerr and Sauer, 2001).

2.2 Lagrangian model

The small-scale cavities are described with a Lagrangian vapour bubble model, i.e. they are tracked individually with the Discrete Bubble Model (DBM). In this approach, the individual bubble dynamics is modelled through the Rayleigh-Plesset equation, in order to consider the collapse and rebound of individual bubbles based on the variations in the surrounding pressure. The Rayleigh-Plesset equation is written as,

$$R(t)\ddot{R}(t) + \frac{3}{2}\dot{R}^2(t) = \frac{p_B(t) - p_l(\infty,t)}{\rho_l} - 4\vartheta_l \frac{\dot{R}(t)}{R(t)} - \frac{2\sigma_{st}}{\rho_l R(t)}, \quad (2)$$

where, R is the instantaneous radius of the bubble , p_l is the surrounding pressure, σ_{st} is the surface tension and p_B is the bubble inside pressure which includes both vapour and dissolved gas pressures. The Lagrangian model has a fourway coupling, i.e., the effects of bubbles on the continuum flow and other bubbles are considered. In this study, the bubbles are decided to interact with each other through collisions. Furthermore, the bubbles may grow (rebound) due to the surrounding pressure variations, and if they become sufficiently large, they are transferred back from the Lagrangian to the Eulerian frame. Also, if a bubble hits a Eularian cavity interface, it will be transferred to the Eulerian.

2.3 Transition Criterion

Transition from the Eulerian to the Lagrangian frame

Cavity structures that are too small to be described using the single fluid mixture approach, are transformed to Lagrangian bubbles. The strategy used to identify this transition is currently similar to the technique used by Vallier (2013). At each time step, the vapour structures are identified with a so-called connected components technique. It consists in associating the adjacent grid cells that have a liquid volume fraction smaller a threshold value (e.g. $\alpha_{th} = 0.9$). The size of the coherent vapour structure is estimated based on the number of cells that they occupy, and small structures are transformed into bubbles. The adjacent cells that fulfil the criterion (i.e., $\alpha_{cell} < \alpha_{th}$) are stored together with the number of the coherent structure (bubbleID) they belong to. A given minimum number of connected cells, denoted by $N_{E\text{-}L}$, is required to represent the smallest vapour structure. Any vapour structure that is described in the Eulerian frame by less than $N_{E\text{-}L}$ cells is a candidate for being transformed to the Lagrangian frame. The position, size and velocity of the new bubbles are extracted from the Eulerian data of the related vapour structure and the liquid volume fraction of the occupied cells is set to 1. The bubbles are small enough to be considered as spherical due to the surface tension and , their diameter is derived from a sphere that has the same volume as the related vapour structure.

Transition from the Lagrangian to the Eulerian frame

A Lagrangian bubble may become very large after coalescence or due to an explosive growth when the surrounding pressure becomes lower than the critical pressure. Bubbles that are too large to be tracked with DBM approach, or large enough to be described through the volume fraction function, should be transformed to the Eulerian framework. The current criterion for this is based on the number of grid cells that the bubble occupies in relation to a threshold value denoted by $N_{L\text{-}E}$. $N_{L\text{-}E}$ should be chosen larger than the $N_{E\text{-}L}$ value so that the bubble is allowed to grow and the model can capture the collapses and rebounds following the growth phase. The bubble may also be transferred to the Eulerian frame if it hit a Eulerian vapour interface. This criterion is measured if the bubble comes close enough to a Eulerian isosurface with $\alpha = 0.5$. When a bubble is transformed to a Eulerian cavity, the cell that hosts the bubble centre and the closest neighbouring cells are filled with vapour. The number of these depends on the bubble volume and also on the available volume in the neighbouring cells that can be converted into vapour.

3 Results

In this section the model performance is validated qualitatively in three simulations.

3.1 Case 1

In this case the collision of two particles in a stationary fluid is simulated. The induced velocity vectors and the particles motions are depicted in Fig. 1. In the left figure, the particles are shown before colliding and in the right figure they are shown afterwards. The induced velocity vectors and

the change in the particles directions after collision are proof of the effect of particles on each other and on the flow field. This case was simulated to test the fourway coupling feature of the solver.

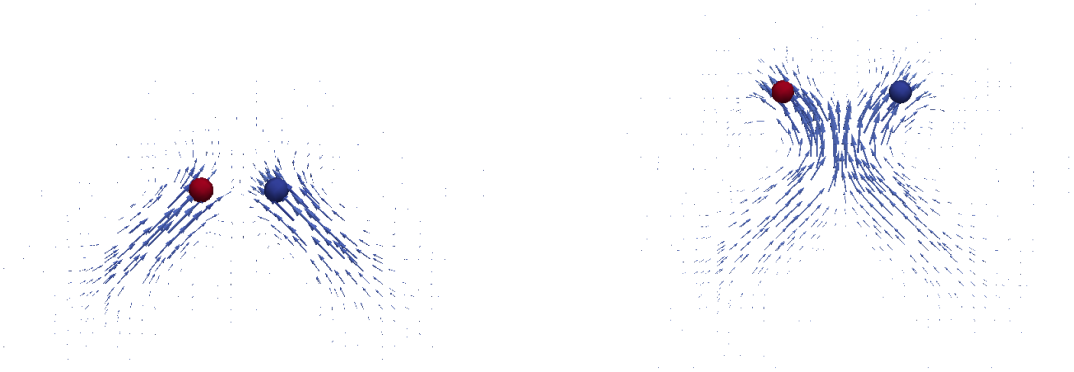

Fig. 1: particle collision in a stationary fluid

3.2 Case 2

In this case a collapsing vapour structure in a 2D channel flow is simulated. The vapour structure is defined by initializing the vapour volume fraction value in the domain and it is subjected to pressure variations along the channel. Some different steps of this simulation are shown in Fig. 2. When the cavity in the first figure becomes small enough, it is transformed into a Lagrangian bubble in the second figure. Due to the dissolved gas content in the bubble and its pressure (which is inversely related to bubble radius) it grows (rebounds) in the third figure. Finally, by further increase in the bubble radius which leads to the reduction of the gas content effect, the bubble collapses again in the last figure. This case was simulated to test the solver capability in capturing the Eulerian-Lagrangian transformation and bubble collapse and rebound.

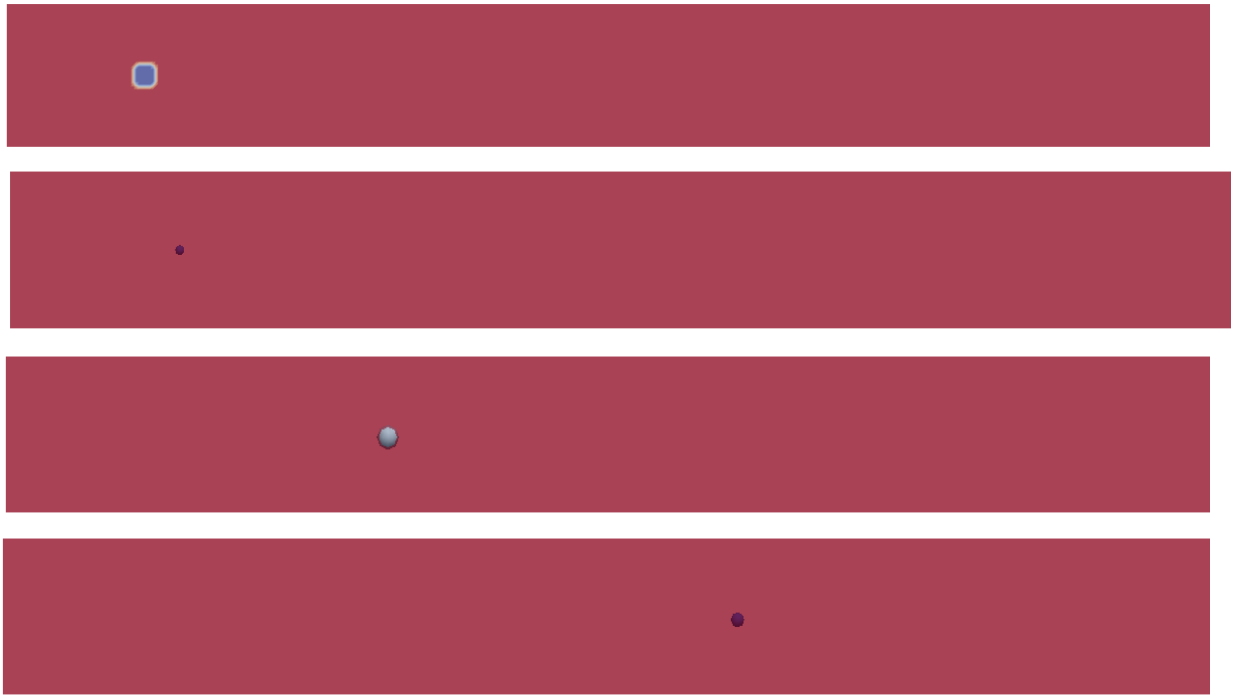

Fig. 2: Cavity collapse and rebound in a 2D channel flow

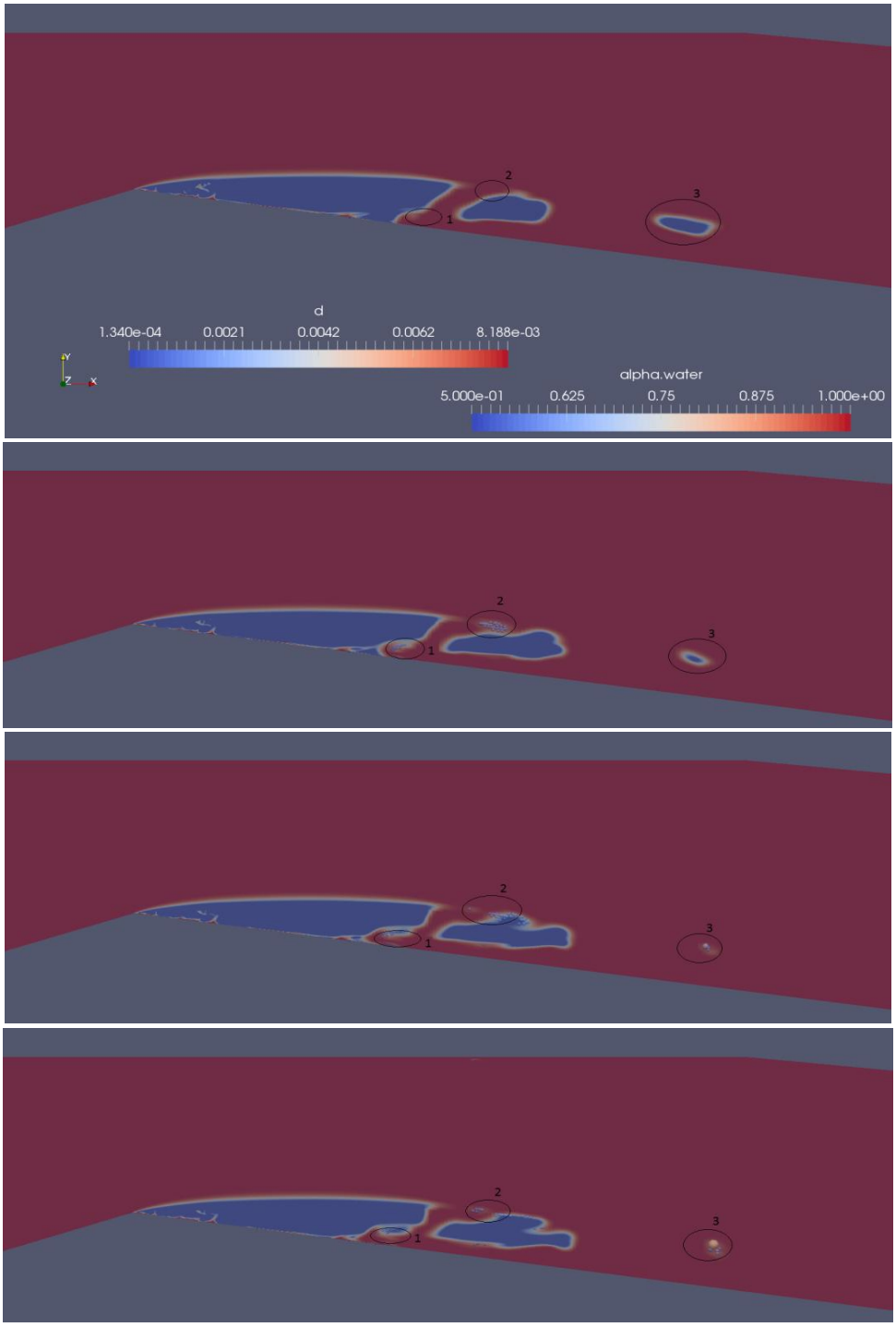

Fig. 3. Cavitating flow in a 2D venturi

3.3 Case 3

In this case a more complicated cavitating flow in a 2D venturi is simulated. Different scenarios can be seen in this simulations, shown in regions 1-3 in Fig. 3. In region 1, some small vapour structures are formed in the re-entrant jet flow which are transformed to small Lagrangian bubbles, and transported upstream. In region 2, the very small cavity structures are transformed to Lagrangian bubbles. Some of these bubbles hit a Eulerian cavity interface in the third figure, and therefore transform back to a Eulerian structure in the fourth figure. Finally, in region 3 a Eulerian cavity is collapsing in the first and second figure. Then, as it becomes small enough, it is transformed to a

Lagrangian bubble in the third figure. Due to the remained vortical flow downstream, some cavities are created which are transformed to more Lagrangian bubbles in the fourth figure. This more complicated simulation clearly show different features of the hybrid solver.

4. Conclusion and future works

In this study, a hybrid solver is described that can resolve an extensive range of scales in cavitating flows. Using this model, bubble growth and collapse due to the variations of the surrounding pressure are captured. The new model has some benefits in erosion prediction as well. In fact the solver can be improved to calculate the radiated acoustic pressure wave due to bubble collapse and rebound which can be used in surface erosion estimation (Eskilsson and Bensow, 2015). Besides that, the residence time of the bubbles gives a less expensive estimation of the regions exposed to successive collapse and rebound.

Although the new model may be used to improve the prediction of cavitating flows, it still needs some basic improvement to give a more reliable estimation. First of all, the Lagrangian library can be improved to include the effect of different forces on bubble trajectory and bubble-wall interaction. Currently, the solver only considers the drag and gravity forces on the bubbles. Also, the bubble-wall interaction is the same as the default settings in the OpenFOAM Lagrangian library which should be improved according to bubble characteristics. Second, bubble-bubble coalescence should be considered in the Lagrangian framework in addition to the currently resolved bubble-bubble collision. Last but not least, there are some more points to be considered like the emitted pressure wave due to the sudden change of the liquid volume fraction in a incompressible flow solver and also to check the continuity criterion during the at the transition times between Eulerian and Lagrangian frames. These last features have not been seen to be considered in the work of Vallier's (2013).

References

M.Abdel-Maksoud, D. Hänel and U. Lantermann (2010) "Modelling and computation of cavitation in vortical flow" . *International Journal of Heat and Fluid Flow*, 31, 1065–1074

R.E. Bensow, G. Bark (2010). "Implicit LES predictions of the cavitating flow on a propeller". *Journal of Fluids Engineering*, 132(4), 041302 .

C. Eskilsson and R.E. Bensow (2015). "Estimation of cavitation erosion intensity using CFD: Numerical comparison of three different methods". *Fourth International Symposium on Marine Propulsors*, 9-16.

M. Gavaises, F. Villa, P. Koukouvinis, M. Marengo and J-P. Franc (2015). "Visualisation and les simulation of cavitation cloud formation and collapse in an axisymmetric geometry". *International Journal of Multiphase Flow*, 68, 14-26.

E. Giannadakis, M. Gavaises, and C. Arcoumanis. (2008). "Modelling of cavitation in diesel injector nozzles". *Journal of Fluid Mechanics*, 616, 153-193.

N.-X. Lu, R.E. Bensow, and G. Bark (2013). "Large Eddy Simulation of Cavitation Developent on Highly Skewed Propellers". *J. Marine Science and Tech*.

C-T Hsiao, J Ma and G. Chahine. (2015). "Simulation of sheet and tip vortex cavitation on a rotating propeller using a multiscale two-phase flow model". *Fourth International Symposium on Marine Propellers*. Austin, Texas, USA.

G. H. Schnerr, I. H. Sezal and S. J. Schmidt (2008). "Numerical investigation of three-dimensional cloud cavitation with special emphasis on collapse induced shock dynamics". *Phisics of Fluids*, 20, 040703.

Vallier, A. (2013). " Simulations of cavitation from the large vapour structures to the small bubble dynamics". *PhD thesis*. Lund University

G. H. Schnerr and J. Sauer (2001). "Physical and Numerical Modeling of Unsteady Cavitation Dynamics". *Proc. 4th International Conference on Multiphase Flow*, New Orleans, U.S.A.

S. Yakubov, T. Maquil and T. Rung (2015). "Experience using pressure-based CFD methods for Euler–Euler simulations of cavitating flows". *Computers & Fluids*, 111, 91–104.