CHALMERS
UNIVERSITY OF TECHNOLOGY

Coupling of the Cavitation Mixture Model with the Discrete Bubble Model

Coupling of the cavitation mixture model with the discrete bubble model

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

1 Introduction

Cavitation is in many cases an undesirable and unavoidable occurrence in industrial hydraulic systems, such as marine propulsion systems and fuel injectors. Cavitation erosion is believed to be the result of violent collapses of the flowing micro-bubbles. Also, cavitation has significant effects on performance and efficiency of the system. Computational Fluid Dynamics (CFD) simulation is an an advancing approach to estimate different characteristics of cavitating flows, and in the last decade it has gained in popularity due to advances in computational resources and modelling. However, due to the various length and time scales present in this complex phenomenon, numerical prediction of cavitation is still a highly challenging task in engineering applications.

Different numerical methods are used to simulate cavitating flows (selectively Bensow and Bark (2010), Giannadakis et al. (2008), Hsiao et al. (2017), Schnerr et al. (2008) and Yakubov et al. (2015)); these models can be categorized in two general approaches. The first approach is based on the mixture equation of state, assuming the thermodynamic equilibrium. As this group of models use compressible solvers, they require very small timesteps to simulate cavitation. Therefore, even if there are suitable models that can adequately estimate the behaviour of cavitation structures, their application in large scale industrial problems is limited, as they require considerably higher computational resources.

The second approach is based on a transport equation for vaporization and condensation. Various numerical models are included in this general classification and the transport equation can be developed in Lagrangian or Eulerian viewpoints. One of the most common Eulerian models is when the flow is treated as a single fluid mixture and mass transfer between the phases is defined by explicit source terms. A limitation to this model (as well as other Eulerian approaches) is that the vapour structures smaller than the grid size, e.g. cavitation nuclei and bubbles, cannot be handled exactly. Also, the mass transfer source term in this model is based on a simplified form of Rayleigh-Plesset equation, in which the cavity inertia is underestimated by neglecting the second derivative of bubble radius (Abdel-Maksud et al., 2010). As cavity inertia becomes more important in the last small scale and fast steps of bubble collapse, this simplification can decrease model accuracy in capturing bubble collapse and rebound. The Lagrangian models, on the other hand, enable more detailed formulations for transport, dynamics and acoustics of discrete vapour bubbles. In this viewpoint the bubble sizes can be much smaller than the grid size and the bubble dynamics is described using a more accurate form of Rayleigh-Plesset equation. However, 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 capturing the small-scale bubbles. There are a few studies in the literature that follows this method, primarily Vallier (2013) and Hsiao et al. (2017). In the current study a multi-scale model 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 bubbles 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 and coupling it with the Lagrangian library. This model is similar to the work of Vallier (2013) but with improvements in some features such as in the treatment of the continuity and volume fraction equations.

In the following sections a more detailed expression of the developed model and a qualitative validation of its performance are presented.

2 Method

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 by solving a transport equation which can be in either a Eulerian or a Lagrangian framework based on the length scale of the structure. One feature of the Eulerian mixture is that it treats the 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. Therefore, to capture small scale cavitation bubbles an extremely high mesh resolution is required. As an alternative less expensive 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. At each timestep, small cavity structures, which cannot be resolved by enough number of computational cells, are transformed to Lagrangian bubbles and the corresponding void fraction of the relative cells (α) is set equal to 1. This transition is shown schematically in Fig. 1 for a simple grid. The grid cells which are occupied by Eulerian cavities are coloured in blue with $\alpha < 1$. Two of the cavities that are resolved by only four cells will be transformed to Lagrangian bubbles. If a bubble becomes large enough afterwards, it is transformed back to a Eulerian structure.

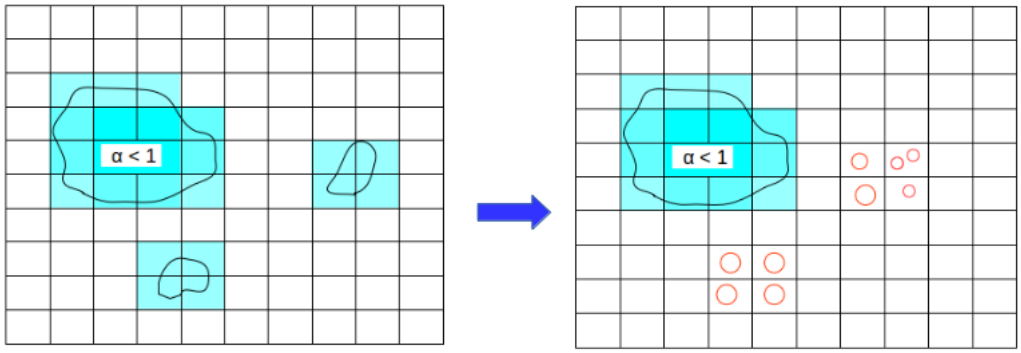

Fig. 1: Transition of small cavities to Lagrangian bubbles

The continuity equation is given by

$$\frac{\partial u_i}{\partial x_i} = (\frac{1}{\rho_l} - \frac{1}{\rho_v})\dot{m}. \tag{1}$$

The equation source term in 1 models the effect of cavity generation on continuity due to condensation, vaporization or cavity transfer. Here, $\dot{m}$ is the rate of mass transfer, and it is here modelled using the well-known Sauer-Schnerr method (Schnerr and Sauer, 2001). Moreover, ρ_l and ρ_v are densities of liquid and vapour.

In the Eulerian formulation a scalar equation is solved for the transport of the liquid volume fraction quantity,

$$\frac{\partial \alpha}{\partial t} + \frac{\partial (\alpha u_i)}{\partial x_i} = \frac{\dot{m}}{\rho_l}, \tag{2}$$

where α is the liquid volume fraction and hence vapour volume fraction equals $1 - \alpha$. The RHS term is the phase change source term.

Finally the flow properties (density and viscosity) formula should be corrected to consider the presence of bubbles in the domain. In the mixture approach the flow density is given by

$$\begin{aligned} \rho_f &= \alpha\rho_l + (1-\alpha)\rho_v, \\ \mu_f &= \alpha\mu_l + (1-\alpha)\mu_v \end{aligned} \tag{3}$$

As mentioned before, when a small cavity structure is replaced by a bubble, the α value should be set to 1 to delete the cavity structure in the Eulerian mixture model. According to the above equation, a sudden change of α can cause a drastic change in the fluid density which leads to spurious pressure pulses

in the domain. To avoid this phenomenon, the bubble contribution should be considered in the density formula (3). This contribution can be specified based on the bubble volume fraction in each grid cell. In other words, the bubble volume fraction should replace the Eulerian cavity volume fraction to avoid drastic changes in flow properties. The cavitation source term in the continuity equation should also be revised in the hybrid solver. Since the Eulerian cavity structures are removed, the continuity source term (Eq. 1) vanishes after transition which may lead to spurious pressure pulses and negative pressures in the flow domain. As a solution to this problem the Lagrangian bubble contribution should be added to continuity source term. In other words, the source term of bubbly cells, $\dot{m}$, should be calculated based on bubble volume fraction rather than the cavity volume fraction. These improvements in continuity and flow property equations are new contributions of the current study.

The Lagrangian bubbles are tracked by solving a set of ordinary differential equations along the bubble trajectory,

$$\begin{aligned} \frac{dx_{b,i}}{dt} &= u_{b,i}, \\ m_b \frac{du_{b,i}}{dt} &= F_d + F_l + F_a + F_p + F_b + F_g. \end{aligned} \tag{4}$$

The RHS of the second equation includes various forces that are exerted on the bubbles which are, from left to right, drag force, lift force, added mass, pressure gradient force, buoyancy force and gravity. Also, the variation of bubble size is calculated by solving the well-known Rayleigh-Plesset equation,

$$R(t)\ddot{R}(t) + \frac{3}{2}\dot{R}^2(t) = \frac{P_B - P_f}{\rho_m} - 4\nu_m \frac{\dot{R}(t)}{R(t)} - \frac{2\sigma_{st}}{\rho_m R(t)}, \tag{5}$$

where R is the bubble radius, P_B is the bubble inside pressure, P_f is the fluid pressure, and σ_{st} is the surface tension.

An important issue with this approach is to consider when bubble size is in the order of grid cell length scale or larger; sometimes a bubble can occupy a number of cells. In OpenFOAM, when a particle hits a wall, the wall boundary condition is applied correctly only if the particle size is smaller than the wall cell height. When a large particle, which occupies several cells, hits a wall the collision of particle with the wall face may not be detected and the particle-wall boundary condition is not applied correctly. Besides that, to solve bubble transport and dynamic equations (Eqs. 4 and 5) the flow properties should be calculated at particle location. In OpenFOAM these parameters are estimated at the center of bubbles. When a bubble diameter is small relative to local grid size, such an estimation is satisfactory; however, when a bubble diameter is in the order of local grid size or larger, calculating flow properties at the bubble center will yield too low accuracy. Therefore, to have a more accurate prediction in bubble transport and dynamics as well as wall boundary condition, a second coarser grid, called Lagrangian grid, is used for bubble tracking. At each timestep, the Eulerian quantities are calculated by solving continuity, Navier-Stokes and Eulerian cavity transport equations in the finer grid. Then, these properties are transformed to the coarser Lagrangian grid to solve bubble transport and dynamic equations. Finally the bubble contributions on the Eulerian flow are calculated and transformed to the finer grid for next timestep. The coupling procedure between the two grids is similar to the utilized one in OpenFOAM *mapFields* function and the mesh to mesh addressing method is *cellVolumeWeight*.

It should be emphasized that the solver is four-way coupling, which means that the effect of bubbles on the flow field as well as bubble-bubble interaction are considered. Bubble-bubble collisions can be of either coalescence or bouncing type based on their diameters and relative velocities. Also the bubble-bubble interaction function is implemented for parallel simulations as well, which means that bubbles in different processors may interact with each other.

3 Results

In this section the performance of the hybrid multi-scale solver in solving a 2D cavitating flow is shown qualitatively. In Fig. 2.a the cavity structures over a 2D NACA0015 hydrofoil is shown. For solving the flow field two grids are generated. The finer grid used for solving the Eulerian equations is composed of

36244 cells while the coarser Lagrangian grid includes only 550 cells. It should be noted that since the Lagrangian cavitation bubbles are present only near the hydrofoil upper surface, the grid cells in other regions can be made quite large to avoid unnecessary computational expenses, resulting in that only 550 grid cells are needed for solving bubble transport and dynamic equations. The small cavity structure in Fig. 2.a should be transformed to Lagrangian bubbles. In Figs. 2.b and 2.c the initial pressure field before transition and the pressure field after transition are depicted respectively. In this case the original governing equations (similar to Vallier, 2013) are used to calculate the flow field. As can be seen, a spurious pressure pulse is generated and the pressure gets negative around the transition region. As mentioned before, this is a result of drastic change in fluid density and sudden change in continuity equation source term. The same flow field is solved using the improved governing equations (as discussed in the previous section) and the resulting pressure field after the transition is shown in Fig. 2.d. As can be seen neither negative pressures nor spurious pressure pulses are generated in this case. It should be noticed that the pressure contours are plotted on a logarithmic scale.

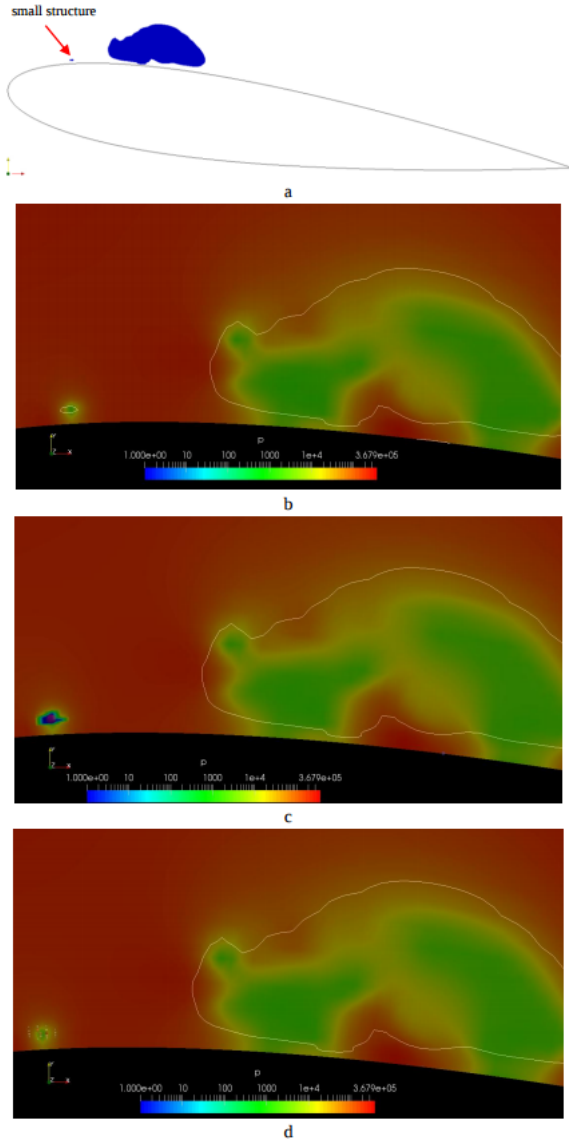

Fig. 2: Transition of small cavities to Lagrangian bubbles

In Fig. 3 different cavity structures including Eulerian cavities and Lagrangian bubbles over the hydrofoil is depicted at 4 different timesteps. Eulerian-Lagrangian transition and evolving of Eulerian structures can be seen in this figure. It should be mentioned that the bubble sizes are enlarged twice to increase the visibility.

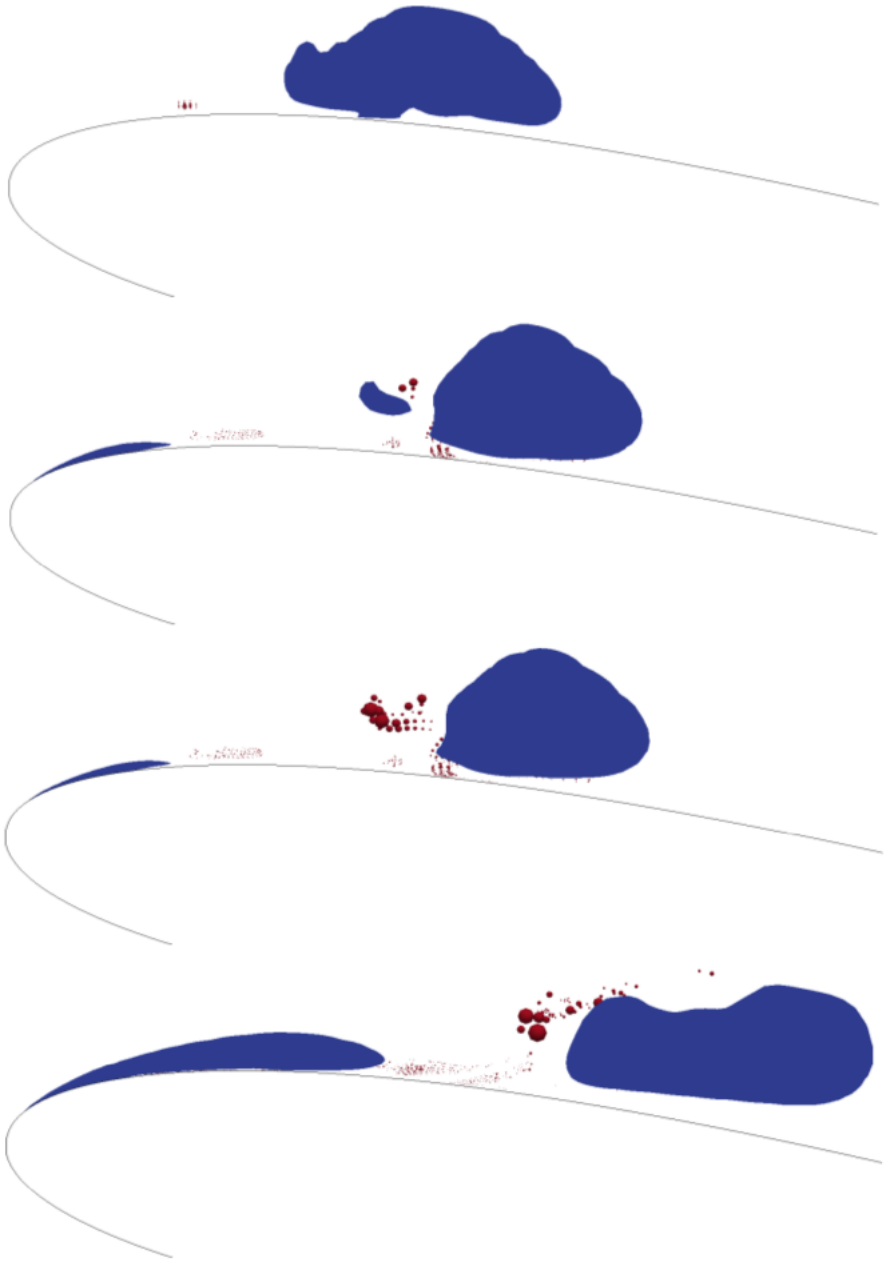

Fig. 3: Eulerian cavities and Lagrangian bubbles on a 2D hydrofoil

4 Conclusion

In this study the negative pressure and spurious pressure pulse problems during Eulerian-Lagrangian transition are fixed. It is shown that this problem is caused by sudden variation of flow properties and source term of continuity equation. However, considering the bubble contribution in Eulerian equations avoids such drastic variations and solves the problem. The hybrid multi-scale solver still needs more improvements to be suitable for simulating complex cavitating flows. In fact, the reverse transition process from Lagrangian bubbles to Eulerian cavities should be corrected so that large groups of bubbles be transformed into Eulerian structures to reduce computational expenses and increase accuracy. Also a more accurate method should be used to overcome OpenFOAM limitations in tracking large bubbles. In this study a second coarser grid was used, however, in a coarser grid some important flow details are missed. In addition to these improvements, the solver performance should be assessed quantitatively as well.

Acknowledgements

This work is funded through CaFE, a Marie Sklodowska-Curie Innovative Training Networks project, grant number 642536.

References

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

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

C-T Hsiao, J Ma and G. Chahine (2017) Multiscale tow-phase flow modeling of sheet and cloud cavitation. *International Journal of Multiphase Flow*, 90, 102-117

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

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

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