



A HYBRID MODEL FOR SIMULATION OF CAVITATING FLOWS

Downloaded from: <https://research.chalmers.se>, 2019-05-21 06:49 UTC

Citation for the original published paper (version of record):

Ghahramani, E., Arabnejad Khanouki, M., Bensow, R. (2017)
A HYBRID MODEL FOR SIMULATION OF CAVITATING FLOWS
12th OpenFOAM® Workshop

N.B. When citing this work, cite the original published paper.

A HYBRID MODEL FOR SIMULATION OF CAVITATING FLOWS

EBRAHIM GHAHRAMANI, MOHAMMAD HOSSEIN ARABNEJAD, RICKARD BENSOW

Mechanics and Maritime Sciences, Chalmers University of Technology,

ebrahim.ghahramani@chalmers.se, mohammad.h.arabnejad@chalmers.se, rickard.bensow@chalmers.se

Keywords: *Cavitating Flow, Disperse Multiphase Flow, Multi-scale Model*

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.

Computational Fluid Dynamics (CFD) simulation, as a supplement or alternative to experimental measurements, can give a more comprehensive understanding of the hydrodynamics of cavitation erosion. Various numerical methods are being used by different researchers today (selectively [1], [2], [3] and [4]); 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. This approach requires very small timesteps since it considers compressible liquid and vapour phases. Therefore, 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 transport equation for vaporization and condensation. This equation can be developed in both of Lagrangian or Eulerian viewpoints. One of the widely used models is when the flow is treated as a single fluid mixture and mass transfer between the phases is defined by explicit source terms. One of the limitations of 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. The Lagrangian models, on the other hand, enable more detailed formulations for transport, dynamics and acoustics of discrete vapour bubbles. 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. In the current study a hybrid 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. The new model is developed by improving the InterPhaseChangeFOAM solver and coupling it with the lagrangian library. This model is similar to the work of Vallier [5] but with improvements in some features including the continuity and volume fraction equations as well as the calculation of mixture properties. This abstract focuses on these improvements.

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

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 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. 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. At each timestep, small cavity structures, that are not 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 Figure 1 for a simple grid. The grid cells that have Eulerian cavities are coloured blue with $\alpha < 1$. Two of the cavities are resolved only by four cells and they are replaced by Lagrangian bubbles. If a bubble becomes large enough afterwards, it is transformed back to a Eulerian structure.

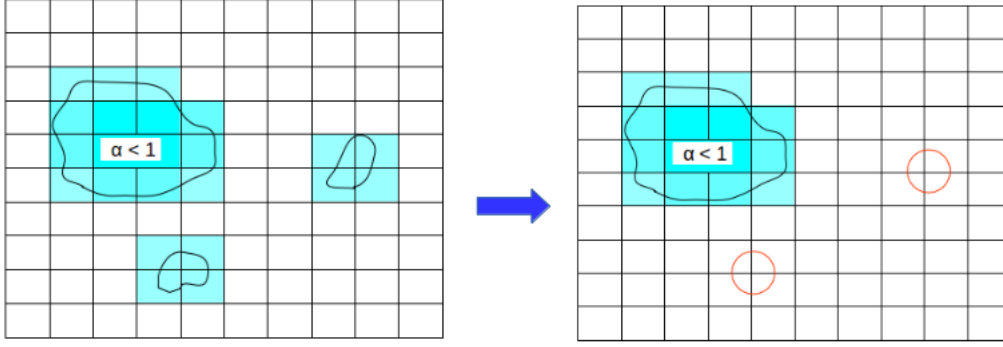


Figure 1: Transition of small cavities to Lagrangian bubbles

The continuity equation is given by

$$\frac{\partial u_i}{\partial x_i} = \left(\frac{1}{\rho_l} - \frac{1}{\rho_v}\right) \dot{m} * \mathbf{pos}(\beta - 1) + \left(\frac{\rho_b}{\rho_v} - \frac{\rho_l}{\rho_v}\right) \frac{D\beta}{Dt}. \quad (1)$$

The first term in the RHS is the effect of cavity generation on continuity which is multiplied by the \mathbf{pos} function. β is the Eulerian fluid contribution in the computational cell and accordingly $(1 - \beta)$ is the bubble contribution. In the computational cells without any bubble, β is equal to 1. Lagrangian bubbles and Eulerian cavity structures do not exist in a computational cell simultaneously. Therefore, when there is a bubble in a cell, the Eulerian cavity effect on the continuity should be set to zero. Since in such a cell, β is less than 1, the \mathbf{pos} factor in the RHS makes the cavity contribution equal to zero. The last term in equation 1 is the bubble effect on the fluid continuity. Also ρ_l , ρ_v and ρ_b are densities of liquid, vapour and bubble.

The Navier-Stokes equation is given by

$$\frac{\partial (\rho_m u_i)}{\partial t} + \frac{\partial (\rho_m u_i u_j)}{\partial x_j} = \frac{\partial \tau_{ij}}{\partial x_j} + \rho_m g_i, \quad (2)$$

where τ_{ij} is the stress tensor and ρ_m is the mixture fluid density.

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

$$\frac{\partial \alpha}{\partial t} + \frac{\partial (\alpha u_i)}{\partial x_i} = \frac{\dot{m}}{\rho_l} * \mathbf{pos}(\beta - 1), \quad (3)$$

where α is the liquid volume fraction and hence vapour volume fraction equals $1 - \alpha$. The RHS term is the phase change source term and \dot{m} is the rate of mass transfer. As mentioned before, Lagrangian bubbles and Eulerian cavity structures do not exist in a computational cell simultaneously and to avoid vapour generation in the cells that are occupied by bubbles, the cavitation source term should be set to zero in those cells and it is considered by adding the \mathbf{pos} function to this term. When there is a bubble in a cell, β is less than 1, therefore the \mathbf{pos} equals zero and no cavity is generated in the cell. This function is also a new contribution in this study. The mass transfer rate (\dot{m}) is obtained using the well-known Sauer-Schnerr method [6] which is already implemented in `interphaseChangeFoam` solver.

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

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

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,

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

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 in this approach which should be considered is the relative sizes of bubbles and grid cells. Sometimes (e.g. when a new bubble is injected) 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. Therefore, to have a correct prediction of wall boundary condition for bubbles, a second coarser grid, which is called Lagrangian grid,

should be used for bubble tracking.

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

$$\rho_f = \alpha\rho_l + (1 - \alpha)\rho_v, \quad (6)$$

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. 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, in this study the density formula (6) is corrected and hence written as

$$\rho_m = \beta\rho_f + (1 - \beta)\rho_b, \quad (7)$$

In the cells without any bubble, $\beta = 1$ and the density is calculated as before. Similarly, the dynamic viscosity is given by

$$\begin{aligned} \mu_f &= \alpha\mu_l + (1 - \alpha)\mu_v, \\ \mu_m &= \beta\mu_f + (1 - \beta)\mu_b \end{aligned} \quad (8)$$

Result

In Figure 2 the cavity structures over the NACA 0015 hydrofoil is shown. The cavitation number is 2.01 with the inlet velocity of 17.3 m/s, the angel of attack is 8° , and the chord length is 0.06 m. As seen in the figure, the large cavities are modelled as Eulerian structure and the small ones are treated as Lagrangian bubbles. Also the corresponding pressure contour on the hydrofoil is depicted in the figure.

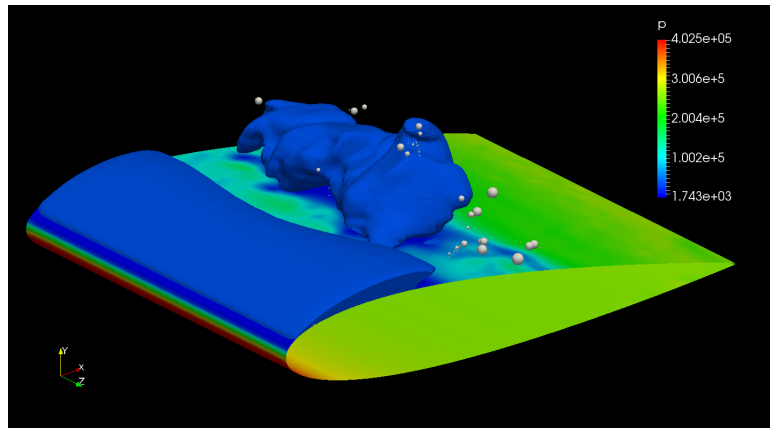


Figure 2: Eulerian cavity structures and Lagrangian bubbles over hydrofoil

Acknowledgments

This work is funded through CaFE, a Marie Skodowska- Curie Innovative Training Networks project, grant number 642536. The computations were performed on resources at Chalmers Centre for Computational Science and Engineering (C3SE) provided by the Swedish National Infrastructure for Computing (SNIC).

References

- [1] R.E. Bensow, G. Bark, *Implicit LES predictions of the cavitating flow on a propeller*. Journal of Fluids Engineering, 2010, 132(4), 041302
- [2] E. Giannadakis, M. Gavaises, and C. Arcoumanis, *Modelling of cavitation in diesel injector nozzles*. Journal of Fluid Mechanics, 2008, 153-193.
- [3] C-T Hsiao, J Ma and G. Chahine, *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, 2015.
- [4] S. Yakubov, T. Maquil and T. Rung, *Experience using pressure-based CFD methods for EulerEuler simulations of cavitating flows*. Computers and Fluids, 2015, 111, 91104.
- [5] A. Vallier, *Simulations of cavitation from the large vapour structures to the small bubble dynamics*. PhD thesis, Lund University, Lund, Sweden, 2013.
- [6] G. H. Schnerr and J. Sauer, *Physical and Numerical Modeling of Unsteady Cavitation Dynamics*. Proc. 4th International Conference on Multiphase Flow, New Orleans, 2001.