Numerical Simulation of Cavitating Flows Using OpenFOAM

Conference Paper · September 2015

CITATIONS
0

READS
631

3 authors:

Abolfazl Asnaghi
Chalmers University of Technology
22 PUBLICATIONS **161** CITATIONS
SEE PROFILE

Andreas Feymark
Chalmers University of Technology
6 PUBLICATIONS **18** CITATIONS
SEE PROFILE

Rickard E. Bensow
Chalmers University of Technology
115 PUBLICATIONS **529** CITATIONS
SEE PROFILE

Numerical Simulation of Cavitating Flows Using OpenFOAM

Abolfazl Asnaghi, Andreas Feymark, Rickard E Bensow, Chalmers University of Technology, Gothenburg/Sweden, {abolfazl.asnaghi,andreas.feymark,rickard.bensow}@chalmers.se

Cavitation is the formation of vapour in a liquid when local static pressure of liquid falls below a critical pressure threshold. As pressure of a larger region gets below the threshold pressure, more liquid will change phase into vapour. Without considering the effects of shear forces in flowing fluids, the pressure threshold is equal to the thermodynamic saturation pressure. This definition, however, has some drawbacks. The thermodynamic saturation pressure is calculated in the conditions where the fluid is steady and in the equilibrium state. Therefore, the rupture of the liquid pocket is just due to the pressure tensile, and effects of shear stress caused by shear velocity are not included. In the previous study by the authors [1], concepts and derivation of cavitation inception by considering the viscous shear stresses were presented, and modification of pressure threshold was addressed. Another modification proposed previously by the authors was to improve the mass transfer rate by considering the velocity strain rate time scale. This modification increases the coupling between the mass transfer rate and the flow local properties, [2].

In the current study, interPhaseChangeFoam solver of OpenFOAM is used to simulate cavitating flows where previously proposed modifications are implemented and analysed in further details. Cavitating flows around NACA0009 and NACA66MOD hydrofoils are simulated to firstly validate the capability of the solver in prediction of cavitation in different operating conditions and secondly to highlight the positive impacts of the proposed modifications on the cavitating flow prediction.

OpenFOAM package, used in this study for numerical simulation, is an open source code written in C++ to model and simulate fluid dynamics and continuum mechanics. It is possible to adopt the code and build new functionalities, libraries, solvers, and utilities. The software is community driven where various communities are working on different fields of applications. This has expedited the progress and development of the software. In OpenFOAM, the spatial discretization is performed using a cell centred collocated finite volume (FV) method for unstructured meshes with arbitrary cell shapes, and a multi-step scheme is used for the time derivatives. To complete the FV-discretization the face fluxes need to be reconstructed from grid variables at adjacent cells, requiring interpolation of the convective fluxes and difference approximations for the inner derivatives of the diffusive fluxes; see [3-5] for more details on the discretization and the numerical schemes used in OpenFOAM.

InterPhaseChangeFoam solver uses Transport Equation Model (TEM) of volume fraction to predict phases' distributions. In this solver, both of the phases are treated as incompressible, isothermal, and immiscible. In order to model the mass transfer between phases, different phase-change models are provided in the solver to simulate cavitation but other mechanisms of phase-change are also supported within this solver framework.

In incompressible TEM approach, the governing equations consist of conservation of mass, and momentum of the effective fluid, and TEM of volume fraction. The filtered equations of mass and momentum can be written as follow,

$$\frac{\partial \rho_m}{\partial t} + \frac{\partial (\rho_m \overline{u_i})}{\partial t} = 0, \tag{1}$$

$$\frac{\partial (\rho_m \overline{u_i})}{\partial t} + \frac{\partial (\rho_m \overline{u_i u_j})}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\overline{S_{ij}} - B_{ij}\right) + \rho_m g_i. \tag{2}$$

In this equation, $B = \rho_m\left(\overline{u_i u_j} - \overline{u_i}\,\overline{u_j}\right)$ is the subgrid stress tensor and $S_{ij} = \mu\left(\partial u_i/\partial x_j - \partial u_j/\partial x_i\right)$ is the shear stress. In Implicit LES (ILES) model used in this study, no explicit function is applied for B; instead the numerical dissipation is considered enough to mimic the action of *B* [6, 7].

The transport equation of liquid volume fraction can be written as follow,

$$\frac{\partial \alpha_l}{\partial t} + \frac{\partial (\alpha_l \overline{u_i})}{\partial x_i} = \frac{\dot{m}}{\rho_l}, \tag{3}$$

α_l is the liquid volume fraction and $\dot{m}$ is the phase change rate between vapour and liquid phases.

Different models have been proposed to represent the phase change rate based on the fluid properties as well as the local flow properties. In the current study, the mass transfer model proposed by Schnerr-Sauer [8] is used and modified according to the proposed modifications.

The Schnerr-Sauer model assumes that there are several vapour bubbles, also called nuclei, inside the liquid which act as the initial sources of the phase change, and cavitation inception starts from their locations and due to their presence. The number, size and distribution of these bubbles can be determined in water quality experiment tests. To simplify the numerical modelling, it is assumed that the initial nuclei have been distributed evenly throughout the liquid, and they have equal size which is the smallest size that vapour bubbles can have. The mass transfer rate between liquid and vapour phases can be defined as follow,

$$\dot{m}_{ac} = C_c \alpha_l \frac{3\rho_v \rho_l}{\rho_m R_B} \sqrt{\frac{2}{3\rho_l}} \sqrt{\frac{1}{|p - p_{threshold}|}} . max(p - p_{threshold}, 0), \tag{4}$$

$$\dot{m}_{av} = C_v (1 + \alpha_{Nuc} - \alpha_l) \frac{3\rho_v \rho_l}{\rho_m R_B} \sqrt{\frac{2}{3\rho_l}} \sqrt{\frac{1}{|p - p_{threshold}|}} . min(p - p_{threshold}, 0), \tag{5}$$

$$\dot{m} = \alpha_l \dot{m}_{av} + (1 - \alpha_l) \dot{m}_{ac} . \tag{6}$$

One of the main issues in the simulation of cavitating flows using TEM is choosing the phase change model coefficients, C_v and C_c, [2]. These coefficients represent the relaxation time that vapour or liquid phase needs to be transferred into the other phase. For instance in the phase change models which are developed based on the bubble dynamics [8, 9], the first derivative of the bubble radius over time represents the relaxation time. However, when the region covered with vapour is larger than one bubble, using this term as the relaxation time may no longer correctly represent the accurate time scale of the phase change making it necessary to modify the coefficients. The chosen values for these coefficients therefore will have direct effects on final results. Another issue which should be taken into account is the dependency of the cavitation characteristics to the local flow properties. Using constant coefficients for mass transfer modelling, or in another word phase change relaxation time, without considering the local properties may impose unrealistic constrains. So, since the cavitation and phase change rate interact locally with the flow properties, one can expect a correlation between flow local time scales and the phase change relaxation time. In previous study [1], we proposed to use the symmetric part of the velocity strain rate for correction of the local phase change relaxation time scale, or in another word modification of the mass transfer coefficients.

Since the modified coefficient is usually bigger than one, using this modification will increase the rate of condensation and evapouration. It has been observed that in cavitating flows, the vapour production coefficient should be large, as high as possible according to [10], to satisfy near instantaneous evapouration. The destruction term, however, allows for some retardation in the condensation [10]. Therefore, the coefficient modification is limited to the evapouration coefficient, Cv where the main flow time scale, $t_\infty = \frac{L_\infty}{U_\infty}$, is employed to normalize the velocity strain rate,

$$C_{v-mod} = C_v \left(1 + t_\infty \left| \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right| \right). \tag{7}$$

In [1, 2], the concept of shear stress effects on liquid rupturing and cavitation inception modification were presented and formulated. As it is stated the pressure threshold that determines the onset of cavitation in the flowing fluid can be expressed as,

$$p_{threshold} = \mu\dot{\gamma} + p_{saturation}, \tag{8}$$

where $\dot{\gamma}$ is shear strain rate. It should be noted that the current study is performed using the Schnerr-Sauer model modified according to these two modifications.

Depending on the local properties of the flow, in the matrix of the discretized volume fraction transport equation, the source term of TEM can become very large compared to the diagonal part due to the very high phase change rate. This may make solving the matrix of discretised equations problematic. In order to improve the solution stability, the source term needs to be rewritten, so the diagonal part can take into account some parts of the source term as an implicit term. The TEM equation can be rewritten by introducing $\dot{V}$ as follow,

$$\dot{V} = \left(\frac{1}{\rho_l} - \alpha_l\left(\frac{1}{\rho_l} - \frac{1}{\rho_v}\right)\right)\dot{m} \tag{9}$$

$$\frac{\partial \alpha_l}{\partial t} + \frac{\partial(\alpha_l \bar{u}_i)}{\partial x_i} = \left(\frac{\partial \bar{u}_i}{\partial x_i} + \dot{V}_v - \dot{V}_c\right)\alpha_l + \dot{V}_c \quad . \tag{10}$$

Here, the source term has been decomposed in two terms. The first term can be treated as an implicit term to increase the stability of solution.

The Schnerr-Sauer phase change model requires that the initial number of nuclei and initial nuclei diameter are predefined to adjust the phase change rate. In order to investigate effects of the phase change model parameters on cavitation behaviour, the cavitating flow around NACA0009 is investigated. The main interest here is to test effects of the number of nuclei and diameter on the cavitation development in a semi-steady flow over a simple geometry. Since the experimental data dose not report time dependency, time averaged numerical results are used for comparison and verification. The details of the experiment can be found in [2]. For numerical simulation, the inlet velocity is set fixed equal to 15m/s, outlet cavitation number is set equal to 0.75, and the foil angle of attack is set equal to 4 degree.

Pressure coefficient distribution over the foil which also presents the cavity size is employed to compare the numerical results with the experiment. In Figure 1, the numerical results using different number of nuclei, 10^6; 10^8; 10^{10} and 10^{12} are presented where the nuclei diameter is kept constant equal to 10^{-5} m. As it can be seen from this figure, using very small value for number of nuclei will lead to under prediction of the cavity. Moreover, at this condition the pressure at the leading edge becomes negative which cannot be correct for the foil with roughness applied on the leading edge. By increasing the number of nuclei, the accuracy of cavity size prediction and pressure distribution increases. For the current condition, it was noticed that increasing the number of nuclei to values higher than 10^{12} does not change the results any further.

Effects of nuclei diameter variation are presented in Figure 2. For these simulations the number of nuclei is kept constant equal to 10^8. This relatively small value for number of nuclei has been selected in order to make the mass transfer rate more sensitive to the variation of nuclei diameter. Increasing the diameter of the nuclei will lead to larger cavity size, and better prediction of the pressure distribution. However, even for very big nuclei diameter, e.g. 10^{-3} m, still the accuracy of the results is very poor. The results also indicate that the Schnerr-Sauer mass transfer rate is more dependent on the number of nuclei than the nuclei diameter.
The quality of water can be measured during the tests, and then the average number of nuclei and diameter can be used as phase change model parameters for simulation. For the cases that the quality

of the water is unknown, however, it can be very laborious to find the appropriate values to obtain reasonable results.

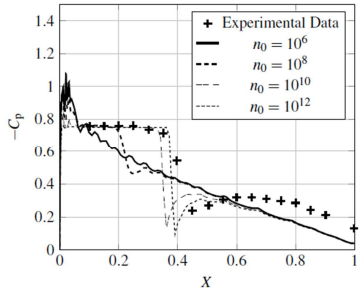

Figure 1. Pressure coefficient distribution of NACA0009 for different number of nuclei

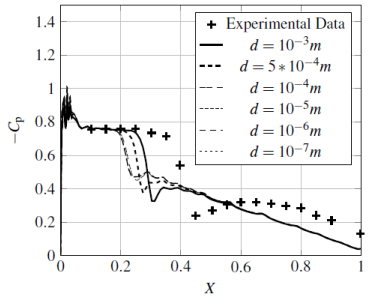

Figure 2. Pressure coefficient distribution of NACA0009 for different nuclei diameter

Since number and diameter of nuclei have direct effects on the mass transfer rate [2], one way to prevent or minimize the negative effects of inappropriate selection of nuclei values is to modify the mass transfer rate coefficients. As it is presented in Figure 3, applying the strain rate modification clearly improves the accuracy of the results. For the results related to the modified coefficient, the pressure at the leading edge is positive which matches with the experiment, and suggests that the mass transfer model is able to predict the vapour generation at the leading edge correctly. Moreover, the simulation is able to predict the sharp pressure variation at the stagnation point next to the cavity closure area, here at the location X=0.4.

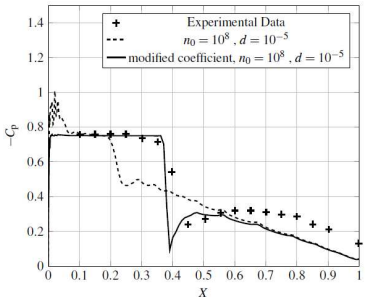

Figure 3. Pressure coefficient comparison between modified and original coefficients of Schnerr-Sauer mass transfer model

The leading edge and mid-chord cavitation on NACA 66 (MOD) hydrofoil is of particular interest for propeller cavitation studies, as it represents the two-dimensional characteristics of propeller blade cavitation. The study here is focus on the mesh resolution effects on the cavitation simulation and force distribution under three different operating conditions. The tested foil has camber ratio of 0.02, thickness ratio of 0.09 and mean line of 0.8 m.

A 2D computational domain is employed for numerical simulation of cavitation around NACA66MOD foil. In the used configuration, the inlet is located 2C upstream of the foil, outlet is located 8C downstream of the foil, and the tunnel width is set equal to 5C, where C is the chord length of the foil. Numerical simulations are carried out for three different cavitation numbers where the angle of attack, 4 degree, and the flow Reynolds number, $2*10^6$, are kept constant. The Reynolds number is calculated based on the chord length and inlet velocity. Three different cavitation numbers have been tested, 0.84, 0.91 and 1.0. The outlet pressure is adjusted to meet these cavitation numbers while the inlet velocity has been kept constant in the simulations.

The computational domain consists of 21000 hexahedral cells which are clustered towards the leading edge, where the pressure gradients are stronger and therefore finer resolution is necessary to capture the flow physics. The cells are also clustered near the foil to provide averaged normal to surface resolution (y^+) equal to 5. More details about the boundary setup and mesh specifications can be found in [2].

In Figure 4 and Figure 5, the pressure coefficient distributions and averaged liquid volume fraction for different cavitation numbers are presented over the foil in the streamwise direction which is normalized by the chord length. Comparing the numerical results with experimental data at the leading edge region indicates that the cavitation has been predicted reasonably well. The main

difference between the numerical results and experimental data is at the end of cavity. There, due to the higher pressure values, condensation is stronger and cavitation simulation will be affected mainly by the accuracy of condensation prediction. At the cavitation number 1.0, the pressure coefficient curve is sharp at the end of the cavity showing clear stagnation point while in the lower cavitation number, e.g. 0.84, the variation of the pressure is smoother. At σ =1.0, the numerical simulation is able to predict the cavitation size and stagnation point very well. At σ = 0.84, the numerical simulation is able to predict the trend well but the discrepancy between the numerical results and experimental data at the end of the cavity is noticeable.

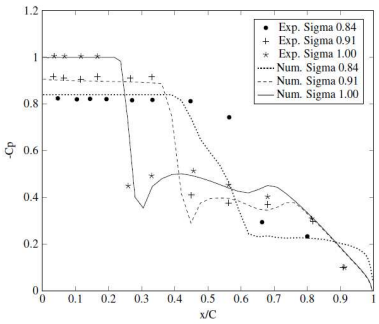

Figure 4. Pressure coefficient of NACA66MOD for different cavitation numbers

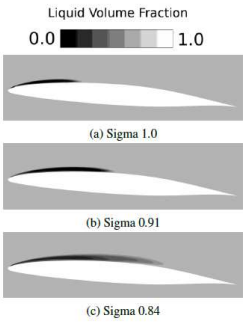

Figure 5. Averaged cavity distribution around NACA66MOD for different cavitation numbers

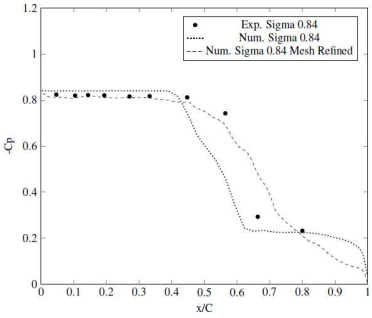

Figure 6. Effects of the mesh resolution at the closure region of the cavity on the pressure distribution, NACA66MOD

As it can be observed from Figure 4, the discrepancy between numerical results and experimental data at the closure region ($0.4 < x/C < 0.8$) is higher for σ = 0.84. Assuming that the transportation of vapour from the leading edge to the closure region is done reasonably acceptable, the pressure distribution at this region is highly dependent on the accuracy of vapour to liquid phase change simulation. Condensation phase change model, compressibility effects, and also numerical settings such as mesh resolution and alpha transportation scheme are some of the parameters that can affect the accuracy of condensation prediction. One possible candidate for the discrepancy between the numerical results and the experimental data is the spatial mesh resolution. As stated previously, the mesh has been clustered towards the leading edge and also trailing edge in order to capture the high flow gradients at those regions. Therefore, the spatial resolution is coarser at the middle region of the foil. By decreasing the cavitation number, the cavity size will increase and gets closer to the middle of the foil which as described has coarser mesh resolution, for instance in σ = 0.84 the closure region is in $0.4 < x/C < 0.6$. In order to investigate the effects of the mesh resolution, the mesh has been refined in the streamwise direction in two steps. At first step the cells inside $0.25 < x/C < 0.9$, and then at the second step the cells inside $0.375 < x/C < 0.8$ are refined. So the cells inside the second box would be four times smaller than the initial cells at that location. In Figure 6, the numerical results of different spatial resolutions are compared with the experimental data for σ = 0.84. As it can be seen from the results, the refinement of the spatial resolution at the closure region has increased the accuracy of prediction of pressure distribution especially at $0.5 < x/C < 0.6$ region. For sheet or semi-steady cavities, it is possible to refine the spatial resolution based on the primary results to get more accurate results later without increasing the computational cost considerably. However, for shedding cavity or vortex cavitation where usually the flow is also three dimensional, providing appropriate spatial resolutions to reasonably model the condensation is not straight forward.

In this paper, numerical simulations of the cavitating flows around the NACA0009 and NACA66MOD foils are computed and compared with experimental data. Schnerr-Sauer mass transfer model and its implementation in OpenFOAM are presented, and effects of the numerical setup (nuclei diameter and number of nuclei) on the final results are investigated. Current results show that considering the velocity strain rate to modify the mass transfer rate will improve the accuracy of the

cavitation prediction. Obtained results also indicate that the cavitation simulation depends on the spatial mesh resolution where lack of appropriate resolution can lead to higher deviation between numerical results and experiment.

Acknowledgements

Financial support of this work has been provided by Rolls-Royce Marine through the University Technology Centre in Computational Hydrodynamics hosted at the Dept. of Shipping and Marine Technology, Chalmers. Computational resources have been provided by Chalmers Centre for Computational Science and Engineering, C3SE.

References

[1] Abolfazl Asnaghi, Andreas Feymark, Rickard Bensow, 'Shear Stress Effects in Cavitating Flows', 17th Numerical Towing Tank Symposium (NuTTS'14)

[2] Abolfazl Asnaghi, 'Developing Computational Methods for Detailed Assessment of Cavitation on Marine Propellers', Licentiate Thesis, Department of Shipping and Marine Technology, Chalmers University of Technology

[3] H. Jasak H.G. Weller, G. Tabor and C. Fureby, 'A tensorial approach to CFD using object oriented techniques'. Computational physics, 12, 1997.

[4] Hrvoje Jasak. 'Error Analysis and Estimation for the Finite Volume Method with Applications to Fluid Flows' PhD thesis, Department of Mechanical Engineering, Imperial College of Science, Technology and Medicine, 1996.

[5] Rusche Henrik. 'Computational Fluid Dynamics of Dispersed Two-Phase Flows at High Phase Fractions', PhD thesis, Department of Mechanical Engineering, University of London, Technology and Medicine, 2002.

[6] Bensow R.E. (2011). 'Capturing Secondary Cavitation - A Step Towards Numerical Assessment of Cavitation Nuisance'. Ship Technology Research 58(2).

[7] Bensow, R.E. & Bark, G. (2010). 'Implicit LES predictions of the cavitating flow on a propeller'. J. Fluids Engng. 132.

[8] J. Sauer and G.H. Schnerr. Unsteady cavitating flow - a new cavitation model based on a modified front capturing method and bubble dynamics. In Fluids Engineering Summer Conf. (FEDSM), 2000.

[9] Sauer, J. (2000). 'Instationäre kaviterende Strömung – Ein neues Modell, basierend auf Front Capturing (VoF) and Blasendynamik'. PhD Thesis, Univ. Karlsruhe.

[10] N Wikstrom, G Bark, and C Fureby. Large eddy simulation of cavitation submerged objects. In 8th Int. conference on numerical ship hydrodynamics, Busan, Korea, 2003.