Cavitation modeling for steady-state CFD simulations

L. Hanimann, L. Mangani, E. Casartelli and M. Widmer
Hochschule Luzern, Technikumstrasse 21, 6048 Horw
E-mail: lucian.hanimann@hslu.ch

Abstract. Cavitation in hydraulic turbomachines is an important phenomenon to be considered for performance predictions. Correct analysis of the cavitation onset and its effect on the flow field while diminishing the pressure level need therefore to be investigated. Even if cavitation often appears as an unsteady phenomenon, the capability to compute it in a steady state formulation for the design and assessment phase in the product development process is very useful for the engineer. In the present paper the development and corresponding application of a steady state CFD solver is presented, based on the open source toolbox OpenFOAM®. In the first part a review of different cavitation models is presented. Adopting the mixture-type cavitation approach, various models are investigated and developed in a steady state CFD RANS solver. Particular attention is given to the coupling between cavitation and turbulence models as well as on the underlying numerical procedure, especially the integration in the pressure-correction step of pressure-based solvers, which plays an important role in the stability of the procedure. The performance of the proposed model is initially assessed on simple cases available in the open literature. In a second step results for different applications are presented, ranging from airfoils to pumps.

1. Introduction
Numerical prediction of cavitating flows and their effects on machine performance and mechanical integrity are a continuously rising field of interest when it comes to the design procedure. Uncertainties already present during experimental tests consequently lead to a broad spectrum of investigation methods. These range from the measurement of the pressure distribution e.g. in 1948 by Rouse and McNown [1] to assess the cavitation state around bluff bodies to more recent methods like computer based image processing in centrifugal pumps [2]. The drawback of these methods was and partly still is the uncertainty of the true volume fraction inside the cavitation region, complicating a thorough investigation with cavitation models. Recent developments as e.g. the X-Ray based measurement techniques [3] aim at closing this gap.
Numerical investigation on the cavitation phenomenon is however still a challenging task. Although a range of simple test cases are available in literature as e.g. the already mentioned cases by Rouse and McNown [1] or simple hydrofoil applications [4], openly accessible turbo machinery applications including trustworthy geometrical data and experimental results are rarely found. One of the best available datasets is the Potsdam Propeller Test Case (PPTC) [5] used during the SMTP symposium [6].
Many authors focus on very detailed numerical investigations of the cavitation phenomenon as single bubble dynamics, time resolved collapse and growth and tracking of particle or bubble
clouds. Most cavitation computations therefore ask for a very fine mesh resolution and are based on transient, i.e. time resolved iterative procedures [7, 8]. While this may be reasonable for e.g. the spray investigation of a diesel injector and the effects on fluid-structure interaction between diesel and injector needle [9], for turbo machinery applications the computational costs would by far exceed the benefits gained from the insights. A reliable and numerically stable formulation is therefore necessary in order to allow the engineer to predict possible occurrence and performance effects of cavitating regions in a steady state analysis. The authors focused therefore on the numerical integration of standard homogeneous mixture-models and their effects on stability and accuracy of the simulation, in order to perform cavitation prediction in a steady state formulation.

The implemented models were validated against measured data and commercial-code results for a hydrofoil test case. The best performing model was then used for the prediction of $\text{HEAD vs. NPSH}_A$ characteristic of an experimentally investigated centrifugal pump impeller at part load.

2. Numerical Methods

In the following section the reader is introduced to the numerical methods used for the implementation of a steady-state cavitation solver. A description of the governing modified RANS and fraction equations as well as underlying cavitation models will be presented.

2.1. Homogeneous mixture model

One of the most common approaches to deal with cavitation is the use of a homogeneous mixture model. It states that the velocity, temperature and pressure between the phases are equal. The justification is given by the assumption that momentum, energy and mass transfer are fast enough to reach equilibrium.

A mixture density (Eq. 1) and viscosity (Eq. 2) are therefore introduced into the Navier-Stokes equations.

\[
\bar{\rho} = \alpha_v \rho_v + \alpha_l \rho_l \\
\bar{\mu} = \alpha_v \mu_v + \alpha_l \mu_l
\] (1)

Subscript $v$ and $l$ are used to indicate vapor or liquid properties respectively. $\alpha$ is the volume fraction of each phase.

2.2. Phase fraction equation

As shown by the above given equations 1 and 2, the influence of varying density and viscosity is solely described by an additional equation for the phase fraction. Different versions of the phase fraction equation are available and depend on the users choice for the conservative flux used for the continuity equation, as will be shown later.

It can either be stated in terms of mass fraction ($f_i$) as given in Eq. 3 or volume fraction ($\alpha_i$) in Eq. 5.

\[
\frac{\delta \bar{\rho} f_i}{\delta t} + \nabla \cdot (\bar{\rho} u f_i) = \nabla \cdot (\gamma \nabla f_i) + R_i
\] (3)

$\bar{\rho}$ is the mixture density introduced in Eq. 1 and $f$ is the mass fraction for phase $i$, with $i$ either $v$ or $l$. The change in mass fraction has to be accounted for by a source term for the mass
exchange rate, here $R_i$. Using the definition of the mass fraction as given in Eq. 3 the fraction equation can be rewritten in terms of volume fraction as given in Eq. 5.

$$f_i = \frac{\alpha_i}{\bar{\rho}}$$  \hspace{1cm} (4)

$$\frac{\delta \rho_i \alpha_i}{\delta t} + \nabla \cdot (\rho_i u_i \alpha_i) = \nabla \cdot (\gamma \nabla \frac{\rho_i}{\bar{\rho}} \alpha_i) + R_i$$  \hspace{1cm} (5)

Equation 5 can be rewritten consistently as Eq. 6 since $\rho_i$ is uniform in the whole domain.

$$\frac{\delta \alpha_i}{\delta t} + \nabla \cdot (u \alpha_i) = \nabla \cdot (\gamma \nabla \alpha_i \frac{\rho_i}{\bar{\rho}}) + \frac{R_i}{\rho_i}$$  \hspace{1cm} (6)

The mass exchange rate $R_i$ is equal to Eq. 7 when expressed for the liquid phase.

$$R_i = R_c - R_e$$  \hspace{1cm} (7)

A negative source contribution is balanced by a decrease of the conserved quantity, i.e. as the mass fraction of liquid decreases more water is evaporating ($e$) than condensating ($c$). $\gamma$ is usually known as the effective exchange rate coefficient. Its exact definition is however not completely clear to the authors since, although often mentioned in the open literature [10, 11, 12], a clear description of the value was not found with the exception of a single paper presented by [13]. It is however written in terms of volume fraction and therefore not directly relatable to the coefficient $\gamma_i$, since the mixture density can not be extracted from the gradient expression, as can be seen from Eq. 8 given by [13].

$$\frac{\delta \rho_e \alpha_v}{\delta t} + \nabla \cdot (\rho_v u \alpha_v) - \nabla \cdot D \nabla \alpha = R_v$$  \hspace{1cm} (8)

The diffusion contribution is defined by Eq. 9

$$D = \bar{\rho} \dot{R} r_e$$  \hspace{1cm} (9)

$\dot{R}$ is the bubble growth radius defined in Eq. 10

$$\dot{R} = \text{sign}(p_{sat} - p) \sqrt{\frac{2}{3} \frac{p_{sat} - p}{\rho_l}}$$  \hspace{1cm} (10)

This is only applied when the domain pressure drops below the saturation pressure, hence inside the cavitation region. This can stabilize the numerical procedure but will diffuse the interface between liquid and vapor phase.

The equivalent radius is defined as:

$$r_e = \left[ \frac{3 \alpha_e V_{cell}}{4 \pi} \right]^\frac{1}{3}$$  \hspace{1cm} (11)

With $V_{cell}$ being the cell volume.

The final forms of equations for a steady state cavitation solver can therefore be stated in terms of mass or volume fraction equation and are given in Eq. 12 and 13 respectively.

$$\nabla \cdot (\bar{\rho} f_i) = \nabla \cdot (\gamma \nabla f_i) + R_i$$  \hspace{1cm} (12)

$$\nabla \cdot (u \alpha_i) = \nabla \cdot \left( \gamma \nabla \alpha_i \frac{\rho_i}{\bar{\rho}} \right) + \frac{R_i}{\rho_i}$$  \hspace{1cm} (13)

Which equation is solved should depend on which variable is conserved in the continuity equation, as will be discussed later.
2.3. The Navier-Stokes Equations

Only one set of Navier-Stokes equations have to be solved, since a mixture approach is used for density and viscosity. The momentum equation is given in Eq. 14. With \( \tau_{i,j} \) being the viscous stresses.

\[
\frac{\delta \bar{\rho} u_i u_j}{\delta x_j} = -\frac{\delta p}{\delta x_i} + \frac{\delta}{\delta x_j} (\tau_{i,j}) + F_{bi}
\] (14)

More important and unclear is however the best practice for the choice of continuity equation. Often a non-conservative, so called non-zero velocity divergence equation is used as given in Eq. 15

\[
\nabla \bar{u} = \left( \frac{1}{\rho_e} - \frac{1}{\rho_l} \right) R_v
\] (15)

This set of equation was suggested by various authors e.g. [14, 15]. Using this equation, the authors suggest to use Eq. 13, since the conserved flux is the divergence of the velocity. The inherent problem of not directly conserving the mass but the volume flow can however be problematic for highly cavitating cases. A new implementation is therefore investigated using a conservative form of continuity equation given in Eq. 16.

\[
\nabla \bar{\rho} \bar{u} = 0
\] (16)

Using this continuity equation the definition of the mass flux is consistent for all equations.

2.4. Solution Procedure

The solution with the final set of equations for steady-state simulations can be summarized as follows:

Momentum Equation

Solve momentum equation for a new velocity field.

\[
\frac{\delta \bar{\rho} u_i u_j}{\delta x_j} = -\frac{\delta p}{\delta x_i} + \frac{\delta}{\delta x_j} (\tau_{i,j}) + F_{bi}
\] (17)

Continuity Equation

Solve continuity equation in pressure or pressure correction form to obtain new pressure and update the velocity fields using e.g. SIMPLE algorithm.

\[
\nabla \bar{\rho} \bar{u} = 0
\] (18)

Mass Fraction Equation

Solve the mass fraction equation to obtain new mass fraction. Recompute the volume fraction with Eq. 4 and recompute the density and viscosity distribution with Eq. 1 and Eq. 2.

\[
\nabla \cdot (\bar{\rho} \bar{u} f_i) = \nabla \cdot (\gamma \nabla f_i) + R_i
\] (19)

Turbulence Equations

For this paper the Menter SST turbulence model was chosen, which adopts an automatic wall treatment in the boundary layer. For consistency it is implemented in its fully compressible version for the convective fluxes.

2.5. Cavitation Models

Four different models are implemented in the in-house modified OpenFOAM® library for the modeling of the mass exchange rate \( R_i \). A proper linearization of the mass exchange rate term to improve the pressure-velocity coupling for steady state predictions was found. While three models are readily available with standard OpenFOAM® distribution, the model by Zwart was added by the authors since it is one of the most common models in commercial codes.
2.5.1. Kunz et al. 2000 [16] The model by Kunz was developed for sheet and super cavitating flows. Sheet cavitation is known to have a gas-liquid interface which is nearly in dynamic equilibrium and pressure and velocity over the interface do not vary heavily [16]. For the liquid-to-vapor change \( R_c \) a simplified Ginzburg-Landau relationship was used.

\[
R_c = \frac{C_{dest} \rho_c \alpha_l \min[0, p - p_{Sat}]}{\rho_l^{\frac{2}{3}} t_\infty}
\]

\[
R_c = \frac{C_{prod} \rho_c \alpha_l^2 (1 - \alpha_l)}{t_\infty}
\]  

(20)

2.5.2. Merkle et al. 2006 [17] The model by Merkle et al. is an improved version of the model by Kunz et al. It uses a boundedness criterion \( f \) for the evaporation and condensation rate to ensure numerical stability.

\[
R_c = \left( \frac{\rho_c \alpha_l}{t_\infty} \right) \min_{f} \left[ 1, \max \left( \frac{p_{Sat} - p}{k_p \rho_v}, 0 \right) \right]
\]

(21)

2.5.3. Schnerr-Sauer-Yuan 2001 [18] This model is based on the approach that the mixture contains a large number of spherical bubbles. The mass exchange rate is then based on a simplified model for bubble growth, based on the Rayleigh-Plesset [19] equation. It should therefore account for non-equilibrium effects.

\[
R_c = \frac{\rho_c \rho_l}{\rho} \alpha_v (1 - \alpha_v) \frac{3}{R_b} \sqrt{\frac{2}{3} \frac{p_{Sat} - p}{\rho_l}}
\]

\[
R_c = \frac{\rho_c \rho_l}{\rho} \alpha_v (1 - \alpha_v) \frac{3}{R_b} \sqrt{\frac{2}{3} \frac{p - p_{Sat}}{\rho_l}}
\]  

(22)

2.5.4. Zwart et al. 2004 [20] Like the Schnerr-Sauer model, the Zwart model is based on a simplified Rayleigh-Plesset equation to account for non-equilibrium effects. In order to better account for the interaction of the cavitation bubbles, the nucleation site density must decrease as the vapor volume-fraction increases. Therefore, the original \( r_v \) was replaced by \( r_{nuc}(1 - r_v) \).

\[
R_c = F_{vap} \frac{3 r_{nuc}(1 - \alpha_v)}{R_b} \rho_v \sqrt{\frac{2}{3} \frac{p_{Sat} - p}{\rho_l}}
\]

\[
R_c = F_{cond} \frac{3 \alpha_v}{R_b} \rho_v \sqrt{\frac{2}{3} \frac{p - p_{Sat}}{\rho_l}}
\]  

(23)

3. Results

Two different test cases have been used for the validation of the implemented cavitation model. The first test case is a NACA 0012 2D profile, the second is a centrifugal pump.
3.1. Naca Profile
The NACA 0012 profile is based on a tutorial case of ANSYS CFX® [4]. The mesh is shown in Fig. 1 and consists of approx. 10000 cells as given from the tutorial. The boundary conditions used for the numerical simulation are the inlet speed of $16.91\frac{m}{s}$ normal to the boundary and a turbulent intensity of 3%. The outlet pressure is $51957 \text{Pa}$. The thermophysical properties of liquid and gaseous water are given in Tab. 1.

|                  | Density $\frac{kg}{m^3}$ | Dynamic Viscosity $\frac{kg}{m s}$ |
|------------------|---------------------------|-------------------------------------|
| Liquid Water     | 997.0                     | $8.899 \cdot 10^{-4}$               |
| Gaseous Water    | 0.02308                   | $9.8626 \cdot 10^{-6}$             |

Table 1: Thermophysical Properties NACA 0012

Various simulations were performed, using two different commercial codes combined with different cavitation models (CC1 Zwart and CC2 Schnerr-Sauer) and the in-house code with all the four models described in Section 2.5. As reference to show the effect of cavitation on the surface pressure distribution, a non-cavitating case is also computed. The major differences to this reference case are found, as expected, on the suction side as a direct result of the cavitation formation.
Differences between solvers and models are mainly in the region behind the cavitation bubble, i.e. in the region where the vapor condensates. In an earlier paper published by the authors [21], the Kunz model was found to performed best, not only due to a more accurate pressure-distribution prediction, but also due to its stable (numerical) behavior. Considering the above results, this conclusion cannot be directly drawn for the NACA 0012 airfoil, since the Zwart model was more accurate in predicting the behavior in the condensing region. In this respect further investigations are necessary, in order to better understand the models behavior for different cases and input parameters.

3.2. Centrifugal Pump Impeller at Part Load Operation

The Zwart model, being the best performer for the airfoil case, was then used for a typical industrial application, i.e. the cavitation prediction in a centrifugal pump. For this purpose a model type centrifugal pump of TU-Braunschweig, located at the Institute of Jet Propulsion and Turbomachinery in Germany was used. Of three different impeller setups available, Impeller A, shown in Fig. 3, was chosen. Experimental data and pictures are extracted, with permission, from the PhD Thesis of Dr. V. Skara [2].

![Figure 3: Impeller Configurations](image)

3.2.1. Numerical setup

The operating conditions are given by a specified flow rate of $138 \times 10^{-3} \text{m}^3/\text{s}$ with an inlet turbulent intensity of 5%. The rotational speed of the impeller is 1200 rpm. The thermophysical properties of the water are given in Tab. The computational domain contains approx. 2.1 mio cells.

|                | Density $\frac{kg}{m^3}$ | Dynamic Viscosity $\frac{kg}{m \cdot s}$ |
|----------------|---------------------------|------------------------------------------|
| Liquid Water   | 998.4                     | $1.001 \times 10^{-3}$                   |
| Gaseous Water  | 0.02308                   | $9.8626 \times 10^{-6}$                  |

Table 2: Thermophysical Properties Impeller A

The goal of the simulation is the prediction of the 3% head drop due to cavitation. This will be observed as a rapid decrease in head of the pump defined in Eq. 24.

$$H = \frac{p_{02} - p_{01}}{\rho g}$$  \hspace{1cm} (24)
Where 2 and 1 always refer to outlet and inlet respectively, with $p_0$ being the total pressure at these locations. This value is then depicted for decreasing $NPSH_A$ (Net Positive Suction Head). The definition is given in Eq. 25.

$$NPSH_A = \frac{p_{01} - p_{Sat}}{\rho g}$$

(Different operating points were reached by reducing the outlet pressure while keeping the saturation pressure on its physical value. Previous solutions with higher back pressures were used as initialisation.

![Figure 4: Head vs NPSH_A](image)

3.2.2. Results

It has to be pointed out that there is a clear mismatch between the measurement data and the simulation results already in the non-cavitating case (High $NPSH_A$ values) as can be seen from Fig. 4. This can be observed for both numerical results (i.e. with the commercial code and with the in-house modified OpenFOAM®-based solver). Main reason for this difference is most likely the material used for the impeller. One of the main investigation purposes in [2] was the visualization of cavitation. Accordingly the impeller was built with Plexiglas to allow the image-based processing of the phenomena. The low stiffness of the material leads to larger deformations compared to the one observed in metal-based test stands. Especially the spacing between impeller and casing was larger than with standard material, leading to an increased leakage flow and therefore additional total pressure loss. The mass flow rate through the impeller is thus further increased due to the additional leakage flow in the rotor cavities. This effect is not taken into account in the CFD computations, since no specific measurement of the leakage flow was performed. The obtained results are therefore affected by this discrepancy between numerical model and experimental setup. Consequently the expected shift toward higher Head and lower NPSH is observed in the CFD results.

Figure 5 shows a comparison between commercial code (left image) and OpenFOAM® computation (right image) using an outlet pressure of 140000Pa, which corresponds to the point in Fig. 4 with the smallest $NPSH_A$ value. As can be demonstrated for the industrial case, the in-house modified version predicts nearly identical results concerning the cavitation region.

4. Conclusions

Detailed implementation details for a steady-state cavitation solver were given, pointing out differences and possibilities in the way of implementation. The implemented models were then
validated on a simple 2D NACA profile and compared against measurements and commercial code results. The presented results showed the significance of cavitation modeling for the prediction of blade loading as can be seen by comparing the suction side pressure coefficient distribution against a CFD computation without cavitation model. In addition, four different cavitation models have been investigated and compared against each other. As mentioned above, further investigation would be necessary to obtain a more clear opinion about which model performs best. It has to be pointed out that besides the quality of the prediction, the influence on numerical stability also has to be judged. From previous experience the Zwart and Kunz model showed the best behavior concerning such a global criterion.

A second test case was then investigated, representing an industrial type application. It is a centrifugal pump impeller at part load which was thoroughly investigated at the TU-Braunschweig. The numerical computation showed discrepancies already in the non-cavitating case against measurement for both OpenFOAM® and commercial code results. This however can be directly related to differences in the leakage flow through the side cavities. This leakage flow was not simulated in the computation as it was not available in the experimental data. However it could be shown, that with the implemented steady-state model on an open source based numerical toolbox good agreement to commercial code could be achieved.

Acknowledgement
The authors would like to thank Dr. V. Skara and Prof. Dr. Ing. J. Friedrichs of IFAS, TU Braunschweig, Germany, for providing the geometry and the system boundary conditions of the investigated centrifugal pump, thus allowing us to test the code capabilities for an industrially relevant test case.

References
[1] Rouse H and McNown J S 1948
[2] Skara V 2014 Experimental Observation of Cavitation Phenomena in Centrifugal Pump Impellers at Part Load Ph.D. thesis TU Braunschweig, Institut fr Flugantriebe und Strmungsmaschinen
[3] Mäkiharju S A 2012 The dynamics of ventilated partial cavities over a wide range of Reynolds numbers and quantitative 2D X-ray densitometry for multiphase flow Ph.D. thesis The University of Michigan
[4] ANSYS C ANSYS CFX Tutorial Tutorial z 313–326
[5] Schiffbau-Versuchsanstalt Potsdam http://www.sva-potsdam.de/pptc_data.html accessed: 2015-12-01
[6] International Symposium on Marine Propulsors http://www.marinepropulsors.com/ accessed: 2015-12-01
[7] Li Z r, Zhang G m, He W and van Terwisga T 2015 A numerical study of unsteady cavitation on a hydrofoil by les and urans method Journal of Physics: Conference Series vol 656 (IOP Publishing) p 012157
[8] Gattoronghi A and Bensow R 2015 Openfoam investigations of cavitation in a flushed water-jet inlet
     Journal of Physics: Conference Series vol 656 (IOP Publishing) p 012059
[9] Örley F, Hickel S, Schmidt S and Adams N 2015 Les of cavitating flow inside a diesel injector including
dynamic needle movement Journal of Physics: Conference Series vol 656 (IOP Publishing) p 012097
[10] Athavale M M, Li H, Jiang Y and Singhal A K 2002 International Journal of Rotating Machinery 8 45–56
[11] Singhal A K, Athavale M M, Li H and Jiang Y 2002 Journal of fluids engineering 124 617–624
[12] Palau-Salvador G, Altozano P G and Arviza-Valverde J 2007 Spanish Journal of Agricultural Research 5
     460–469
[13] Srinivasan V, Salazar A J and Saito K 2009 Applied Mathematical Modelling 33 1529–1559
[14] Sauer J and Schnerr G 2000 Unsteady cavitating flow—a new cavitation model based on a modified front
capturing method and bubble dynamics Proceedings of 2000 ASME Fluid Engineering Summer Conference
     pp 11–15
[15] Spalding D 1974 A method for computing steady and unsteady flows possessing discontinuities of density
     CHAM Report vol 910/2
[16] Kunz R F, Boger D A, Stinebring D R, Chyczewski T S, Lindau J W, Gibeling H J, Venkateswaran S and
     Govindan T 2000 Computers & Fluids 29 849–875
[17] Merkle C and Li D 2006 Sankaran, Multi-Disciplinary Computational Analysis in Propulsion (AIAA-2006-
     4374, 42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference, Sacramento, USA)
[18] Yuan W, Sauer J and Schnerr G H 2001 Mécanique & industries 2 383–394
[19] Rayleigh L 1917 The London, Edinburgh, and Dublin Philosophical Magazine and Journal of Science 34
     94–98
[20] Zwart P J, Gerber A G and Belamri T 2004 A two-phase flow model for predicting cavitation dynamics
     Proceedings of the fifth international conference on multiphase flow, Yokohama, Japan
[21] Hanimann L, Mangani L, Casartelli E and Widmer M 2016 Steady-state cavitation modeling in an open
     source framework: Theory and applied cases ISROMAC 2016, Symposium on Rotating Machinery