An Evaluation of CFD Cavitation Models using Streamline Data

Michael P. Kinzel[1], Jules W. Lindau[1], and Robert F. Kunz[2]

The Pennsylvania State University, University Park, PA, USA
[1] *Applied Research Laboratory* [2] *Department of Mechanical Engineering*

Abstract
In the present work, finite-rate cavitation models common to multi-phase computational fluid dynamics (CFD) are evaluated. These evaluations are based along streamlines extracted from benchmarked CFD. The results include comparative studies of: (1) full Rayleigh-Plesset equation (RPE), (2) simplifications to the RPE, and (3) other cavitation models. Additionally, using this approach, numerical uncertainty is evaluated at a much higher level than previously considered. In the context of developed cavitation, the present assessments elucidate similarities and differences between cavitation models and suggest mesh requirements are more demanding the normally considered.

Introduction

Cavitation models used in CFD stem from a range of physical assumptions and can be roughly categorized into thermodynamic state or bubble dynamics modeling. The present work focuses on the latter, which spans full bubble dynamics and finite-rate cavitation models. Both approaches model rates of nuclei growth and bubble collapse. Finite-rate cavitation models are common practice for multiphase CFD model formulations. Examples of widely used forms include those developed by Merkle *et al.*[1], Kunz *et al.* [2], Sauer and Schnerr [3], Singhal *et al.*[4], Zwart *et al.*[5] and others. One distinction of the models of Singhal, Sauer, and Zwart, with respect to the comparatively ad-hoc models of Merkle and Kunz, is that the former are approximate forms to the RPE. In other words, many finite-rate models for cavity growth and collapse are based on a simplified RPE. Despite their more physical grounding, such approximate RPE models have not displayed more accurate results.

In the context of developed cavitation, the present effort aims to improve the understanding of cavitation models. The analysis is based on benchmarked cavitating CFD results for the cavitating flow over an axisymmetric head form of varying cavitation numbers ($\sigma = \frac{p_\infty - p_{sat}}{0.5\rho_l V_\infty^2}$). Rather than evaluating the fully-coupled system of equations on the entire domain, the process is simplified to an ordinary differential equation (ODE) modeling nuclei growth and collapse along streamlines extracted from the benchmarked CFD solutions. Thus, the CFD pressure fields on the streamlines are used as time-varying forcing functions for the cavitation model formulated as an ODE. Such an approach has the advantage of evaluating and comparing cavitation models in the context of a validated CFD flow field, while avoiding complications and costs associated with the full domain and equation set. Such efforts aim to better understand various models and computational mesh requirements for predicting cavitation using multiphase CFD.

Methods

Cavitation Modeling

Finite-rate cavitation models are often cast into a vapor mass conservation equation form that uses source terms to model gas formation and destruction processes. Such a vapor-mass conservation equation is given by [1]:

$$\frac{\partial \rho_v \alpha_v}{\partial t} + \frac{\partial \rho_v \alpha_v u_i}{\partial x_i} = \dot{S}^+ - \dot{S}^- \tag{1}$$

Here, ρ_v is the vapor density, α_v is the vapor volume fraction, and the source terms, $\dot{S}^+$and $\dot{S}^-$, model bubble growth and collapse, respectively. When combined with a Navier-Stokes-based solver, via fluid properties, this yields a well-established cavitation model approach for CFD [1, 2, 4, 3]. As previously

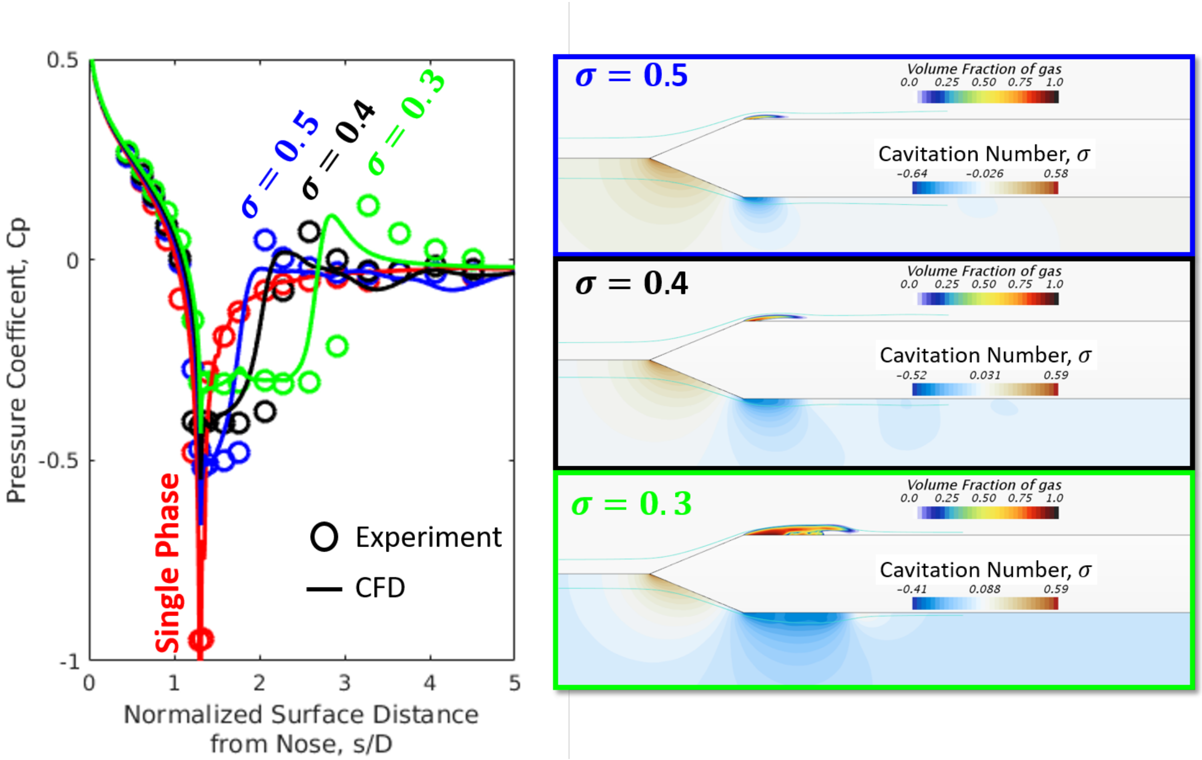


Figure 1: Benchmark comparing CFD to Rouse and Mcknown experiments measuring cavitation over a conical-shaped head form. The left plot indicates C_P predictions (lines) versus experimental measurements (symbols). The contour plots to the left are flow visualizations (upper plots are vapor volume fractions, lower plots are C_P).

mentioned, several cavitation models are formed using an approximate RPE. The full RPE, in an Eulerian reference frame consistent with CFD models, is given as

$$R\frac{D^2R}{Dt^2} + \frac{3}{2}\left(\frac{DR}{Dt}\right)^2 + \frac{4\nu_L}{R}\frac{DR}{Dt} + \frac{2S}{\rho_L R} = \frac{p_v - p}{\rho_L} + \frac{p_{G_0}}{\rho_L}\left(\frac{R_0}{R}\right)^{3\gamma}. \tag{2}$$

Here we discuss the model of Singhal *et al.* [4], with the turbulence terms neglected, providing source terms that approximate the RPE as

$$\dot{S}^+ = F_{vap}\frac{\rho_v\rho_l\alpha_l}{\sigma}\sqrt{\frac{2}{3}\frac{max(p - p_{sat}, 0)}{\rho_l}} \tag{3}$$

and

$$\dot{S}^- = F_{cond}\frac{\rho_v\rho_l\alpha_v}{\sigma}\sqrt{\frac{2}{3}\frac{max(p_{sat} - p, 0)}{\rho_l}}. \tag{4}$$

Note that the model clearly displays the Rayleigh approximation to the RPE, i.e. $\frac{dR}{dt} = \sqrt{\frac{2}{3}\frac{p-p_{sat}}{\rho_l}}$. The remaining constants and variables indicate how the model deviates from the Rayleigh approximation (i.e., empirical corrections). Similar to the Singhal model [4], the most of the ubiquitous finite-rate cavitation models may be cast algebraically into a similar form and can be solved as a first-order ODE in time[6]. The Singhal model is applied in the baseline CFD solution to generate streamlines for ODE integration of the Singhal, Sauer, Kunz, and Rayleigh models.

CFD Model

The basis of this effort rely on benchmarked CFD results obtained using an implementation of the Singhal model [6] into Star-CCM+ 12.06 [7]. The model relies on the segregated-flow solver in Star-CCM+ that

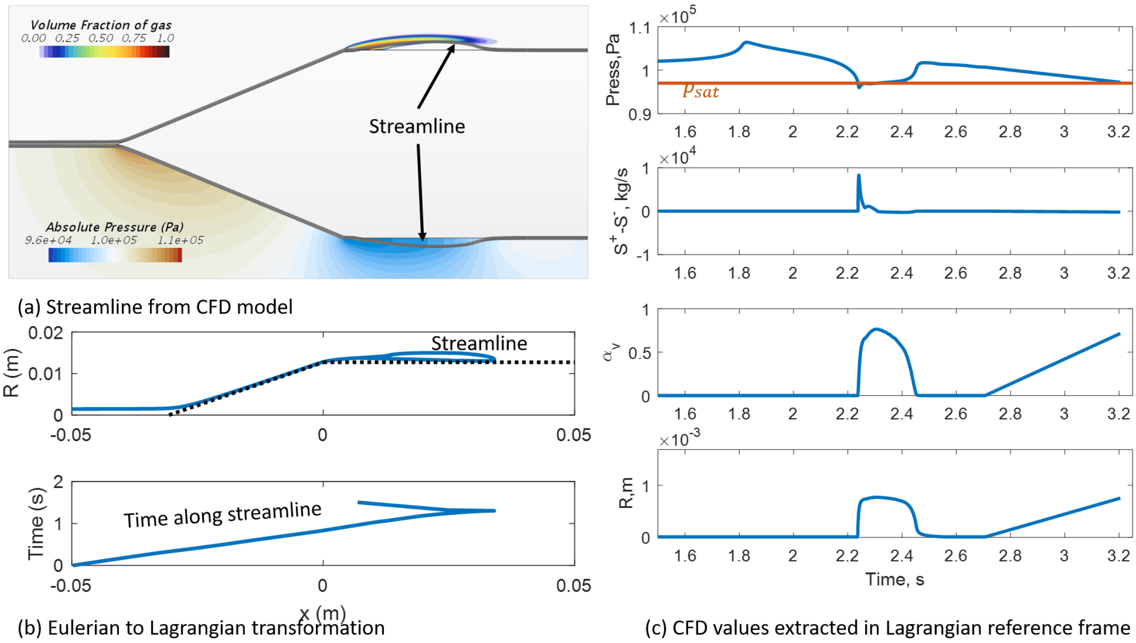


Figure 2: Description of the conversion to a Lagrangian reference frame. The plots display the streamline extraction process to convert to ODE evaluations of cavitation models.

conserves global mass, momentum, and species mass. The numerical scheme is formally second-order accurate in space and the simulations are run using a steady solution approach. A Reynolds Averaged Navier-Stokes (RANS) turbulence model is used based on the $k-\epsilon$ turbulence model in the context of the models discussed in Kinzel *et al.*[8]. Note that the present analyses are expected to be independent of turbulence model and findings should extend to any turbulence model choice.

The physical properties used in this modeling effort are as follows. The model is based on an incompressible fluids with a liquid water phase ($\rho_l = 1000kg/m^3$,$\mu_l = 0.001Pa-s$) and a gaseous phase(with properties similar to air, $\rho_v = 1.2kg/m^3$, $\mu_v = 1.85\times10^{-5}Pa-s$). In addition, in the context of the unified model presented in Kinzel et. al [6], the model uses a free stream nuclei with a site density of $400,000m^{-3}$ and a radius, R_0, of $10\mu m$. Lastly, the saturated vapor pressure, p_{sat}, is specified as $p_\infty - \sigma 0.5\rho_l V_\infty^2$ where p_∞ is 1 atmosphere. These properties define the physical inputs relevant to the presented results.

Results from the aforementioned CFD model are compared to experimental measurements of surface pressure from a cavitating flow over a conical-shaped head form from Rouse and Mcknown [9]. The benchmarking exercise is summarized in Fig. 1. In the left plot, are comparisons of the pressure coefficients, i.e.,$C_P = \frac{p-p_\infty}{0.5\rho_l V_\infty^2}$, measured (symbols) to the CFD predictions (lines) at various cavitation numbers. The overall correlation between CFD and experiment is reasonable, hence, extracting streamlines and pressure data are a reasonable starting point to assess cavitation models.

Streamline Analyses

Enabling cavitation-model evaluation using ODE solutions demands that the CFD streamline data are converted to the Lagrangian reference frame. The process demands temporal data on the nuclei as the transport along the streamline, hence, time along the streamline must be computed as

$$t_p = \int_0^p \frac{ds}{|V_p|}. \tag{5}$$

Here, p is a point on the streamline at time t_p that is relative to the streamline start time. Additionally, the velocity magnitude, $|V_p|$, and a differential distance ds, i.e., the differential distance along the

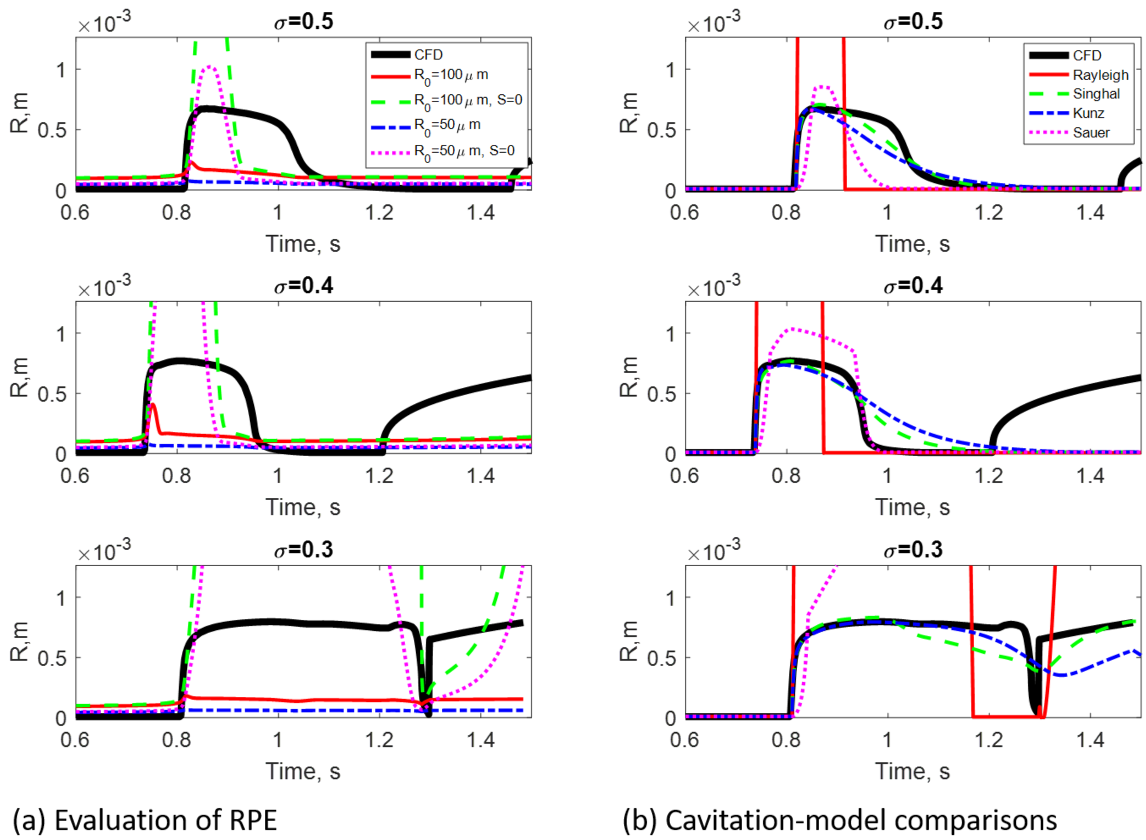


Figure 3: Evaluation of the various cavitation models along a streamline. Part (a) indicates the RPE solutions whereas part m(b) compares results from various finite-rate models.

streamline, are required. Sample results and extraction of the data are depicted in Fig. 2. The pressure and velocity are extracted along the streamline providing the necessary data for the transformation and the ODE forcing function (i.e., the pressure-time curve indicated the upper-right plot in Fig. 2). Such a pressure history drives the cavitation model, which can be compared to the CFD cavitation model, vapor content, and effective radius in the second through forth plots to the right of Fig. 2. Note that the CFD radius is computed using an approximation of: $R = \left(\frac{\alpha_v}{4/3\pi N_b}\right)^{1/3}$.

Results and Discussion

We now compare various cavitation models to the RPE and evaluate mesh dependencies associated with cavity gas content. Results are expected to yield insight into cavitation models, their performance, and paths to improvement.

RPE Solution

First, consider the application of the RPE in the context of these developed cavitation cases. Results are plotted in Fig. 3 (a) for three different σ values and are compared to CFD Singhal-model results. Note that these RPE results assume free-stream nuclei having radii of $R_0 = 50$ or $100\mu m$, which are both evaluated with and without surface tension. Results indicate that the full RPE, with surface tension and exceedingly large nuclei 10× larger than assumed in the CFD model, do not correspond to the benchmarked CFD. Specifically, the RPE predicts that nuclei growth is minimal when passing into an already ruptured cavity void. Alternatively, and similar to finite-rate model formulations, neglecting surface tension indicates that the RPE predicts cavitation similar to the CFD. Such results are not intended to suggest that full RPE is an invalid model. It does, however, suggest that the RPE is not directly applicable in the present analysis. We hypothesize that the full RPE may not be directly applicable to developed cavitation due to it not considering vapor addition.

Cavitation Model Comparison

Now consider comparing various cavitation models along these streamlines. This may be considered an extension of the efforts from Kinzel *et al.* [8] that compared cavitation models using artificial pressure distributions. In Fig. 3 (b), are comparisons of results from the (1) Rayleigh, (2) Singhal, (3) Kunz, and (4) Sauer models for three σ values. Recall that the reference CFD result was generated with the Singhal model.

First notice the discrepancies between the CFD- (CFD) and ODE-Singhal (Singhal) models in Fig. 3 (b). The observed discrepancies are expected to be a result of the ODE-model treating cavitation processes as a pure advection process, which neglects the vapor diffusion, limiters, and other numerical factors present in the CFD. In general, the model results are comparable suggesting verifying that the streamline method can yield insight.

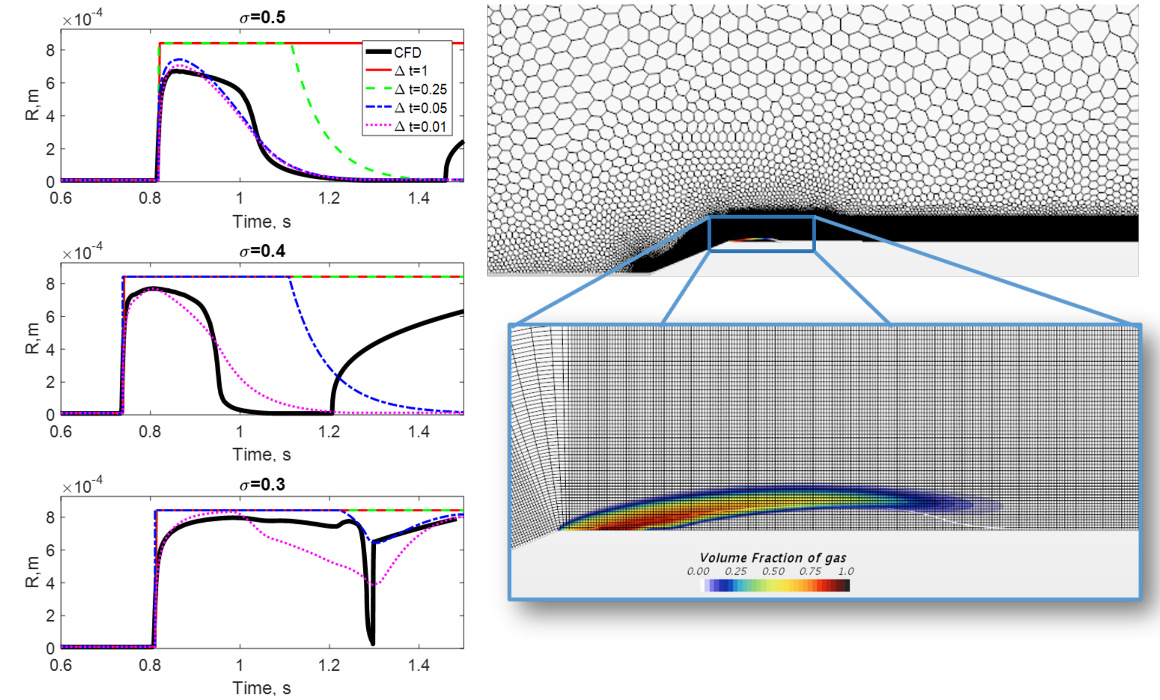
Model-to-model comparisons in Fig. 3 (b) suggests good agreeable between the cavitation models. Note that, for each model, the cavitation constants are consistent across all σ values. The results form the Singhal- and Kunz-ODE (Kunz) models correlate well with the Kunz model constants set to $C_{evap} = 2.0$, $C_{e,ref}$, and $C_{cond,1} = 1.25C_{c,ref}$. This is a slight modification from the analysis[6] suggesting $C_{evap} = 1.15$, $C_{e,ref}$, and $C_{cond,1} = 10.0C_{c,ref}$. The Singhal and Kunz models do deviate from the Sauer model, and when compared to the RPE, the the Sauer model better correlates with the RPE model without surface tension. Note that the Sauer model constants deviated significantly from those predicted in Kinzel *et al.*[6]. Specifically, the present Sauer-model results use $C_{evap} = \frac{\rho_l}{40{,}000\alpha_{v,0}}C_{e,ref}$, and $C_{cond,2} = 0.0065\frac{\rho_l^2}{\rho_v}C_{c,ref}$ as compared to the previous analysis[6] suggesting $C_{evap} = 1.15C_{e,ref}$, and $C_{cond,2} = \frac{\rho_l^2}{\rho_v}C_{c,ref}$. This indicates that the previous analysis[6] should be refined for the Sauer model. Lastly, the Rayleigh model indicates much faster evaporation and condensation rates as the model indicates it models too rapid of growth. Overall, the general character of the Singhal and Kunz models are very similar, some deviation is observed with the Sauer model, and the Rayleigh model indicates a much more violent cavitation process.

Effect of Mesh

The next analyses indicates that cavitation prediction has a strong mesh resolution dependency that is exacerbated with lower σ. The analysis focuses on the Singhal-ODE solution and only focuses on cavitation generation (which does not necessarily correlate to loads). In Fig. 4 (a), the results with different time-step sizes are compared with respect to refining from a Δt of unity which corresponds to the CFD mesh (depicted in Fig. 4 (b)). Such a mesh is considered to be a standard cavitation resolution. Refined resolutions are obtained through ODE integration to shorter times steps. In evaluating Fig. 4 (a), it is clear that cavity generation is highly dependent on the resolution. Such an observation indirectly relates to CFD mesh resolution suggesting CFD meshes may need to be significantly refined to evaluate cavitation dynamics.

Summary

In this work, flow-field data extracted on streamlines from a CFD solution are used to evaluate cavitation models. In comparing CFD results to results from such streamline analyses, a reasonable correlation was observed indicating the usefulness of the approach and greatly simplifying model comparisons. When cavitation models are analyzed, the present results suggest strong similarity in various cavitation models. However, there was one exception. The full RPE with surface tension deviated significantly indicating that, for the given pressure fields, nuclei could not effectively supply gas volume for these developed cavitation conditions. This may suggest that nuclei expansion is not an important process for developed cavitation. In addition, we also observed a significant sensitivity in the development of a cavity associated with the computational mesh, suggesting that numerical uncertainty is large cavitation models. Such insights are useful in that they provide insight to improve cavitation models.



(a) Effective mesh sensitivity

(b) Computational mesh and detail near cavity

Figure 4: Evaluation of the discretization sensitivity. Discretization sensitivities are presented in part (a) where a Δt of unity corresponds to the CFD computational mesh used provided in part (b).

References

[1] Charles L Merkle, J Feng, and Phillip E O Buelow. Computational modeling of the dynamics of sheet cavitation. In *3rd International symposium on cavitation, Grenoble, France*, volume 2, pages 47–54, 1998.

[2] Robert F. Kunz, David A. Boger, David R. Stinebring, Thomas S. Chyczewski, Jules W. Lindau, Howard J. Gibeling, Sankaran Venkateswaran, and T. R. Govindan. Preconditioned Navier-Stokes method for two-phase flows with application to cavitation prediction. *Computers & Fluids*, 29(8):849–875, 2000.

[3] Jürgen Sauer and G. H. Schnerr. Development of a new cavitation model based on bubble dynamics. *Zeitschrift für Angewandte Mathematik und Mechanik*, 81(S3):561–562, 2001.

[4] Ashok K Singhal, Mahesh M Athavale, Huiying Li, and Yu Jiang. Mathematical Basis and Validation of the Full Cavitation Model. *Journal of Fluids Engineering*, 124(3):617–624, 8 2002.

[5] Philip J Zwart, Andrew G Gerber, and Thabet Belamri. A Two-Phase Flow Model for Predicting Cavitation Dynamics. In *ICMF 2004 International Conference on Multiphase Flow*, 2004.

[6] Michael P Kinzel, Jules W Lindau, and Robert Kunz. A Unified Model for Cavitation. In *FEDSM17 2017 ASME*, Waikoloa, Hawaii, USA, 0. ASME.

[7] Siemens. Star CCM+ 12.06, 2017.

[8] M P Kinzel, R F Kunz, and J W Lindau. An Assessment of CFD Cavitation Models Using Bubble Growth Theory and Bubble Transport Modeling. In *International Symposium on Transport Phenomena and Dynamics of Rotating Machinery (ISROMAC 2017)*, Maui, Hawaii, USA, 2017.

[9] Hunter Rouse and John McNown. Cavitation and pressure distribution: head forms at zero angle of yaw. *University of Iowa Studies in Engineering*, 1 1948.