Application of Homogeneous and Inhomogeneous Two-Phase Models to a Cavitating Tip Leakage Vortex on a NACA0009 Hydrofoil

[1]Jonas Wack*; [1]Stefan Riedelbauch

[1]*University of Stuttgart, Germany*

Abstract

Two-phase modeling plays an important role for the correct prediction of cavitation phenomena. However, the computational effort grows with increasing complexity of the model. Consequently, complex models should only be applied if they lead to a significant improvement of the results. In the scope of this study, homogeneous and inhomogeneous models are applied to a cavitating tip leakage vortex on a NACA0009 hydrofoil at 10° incidence angle. Two different cavitation model constants from literature are applied for the homogeneous model. For the inhomogeneous approach, the impact of bubble size diameter is investigated. Furthermore, a constant drag coefficient 0.44 is compared to Schiller Naumann drag model. The results indicate that for the presented case, the inhomogeneous model does not lead to a significant improvement of the simulation results, which can be explained by a low Stokes number due to small bubble diameters. All investigated setups show similar results, which underline that the homogeneous model is sufficient to resolve the present cloud cavitation as well as the cavitating tip separation and tip leakage vortex.

Keywords: cavitation; two-fluid model; tip leakage vortex

Introduction

Owing to the integration of other renewable energies into the electrical grid, hydro power plants are more and more operated at off-design conditions. This comes across with secondary flow phenomena and consequences like cavitation have to be considered. In a water turbine, several different forms of cavitation can occur and coexist at the same time. For instance, in a Kaplan turbine a cavitating tip leakage vortex, sheet or cloud cavitation on the runner blades and a cavitating draft tube vortex rope can occur simultaneously. The occurrence of different forms of cavitation makes it challenging to select a suitable two-phase modeling approach.

For cavitation simulations it is common practice to use a model – called homogeneous model – that treats the two phases as one mixture. However, cavitation simulations on a Francis turbine at deep part load conditions showed shortcomings to correctly reproduce the simulated cavitation appearance compared to measurements [1]. This raises the question whether the more complex inhomogeneous two-phase modeling approach, also called two-fluid model, can increase accuracy of simulation results. In the scope of this study, the two approaches are applied to a NACA0009 hydrofoil that is facing a cavitating tip leakage vortex, which is generated by the flow through the gap between hydrofoil side wall and cavitation tunnel wall. This test case has been measured in detail by Dreyer et al. [2, 3, 4]. Numerical studies have been performed by Decaix et al. [5] who showed that on a coarse mesh the cavitation volume is significantly underestimated.

Numerical modeling

The commercial code Ansys CFX version 17.2 is used for CFD simulations. In the first step, single phase simulations are performed for a mesh study and the selection of suitable turbulence models. For the two-phase simulations, the homogeneous and inhomogeneous models are applied as multiphase modeling approach.

The more complex inhomogeneous model consists of one set of governing equations (mass and momentum conservation) per phase, which results in a significantly increased computational effort compared to single phase simulations. This modeling approach allows separate velocity fields for liquid and vapor phase, which is the cause of interfacial forces like drag force. If the relative motion between the phases is neglected, the mixture can be treated as a pseudo-fluid. This results in the homogeneous model, where only one set of governing equations needs to be solved for the mixture phase. Additionally, a transport equation for the volume fraction α has to be solved. All in all, the computational effort of the homogeneous model is significantly lower compared to the inhomogeneous model.

*Corresponding Author, Jonas Wack: jonas.wack@ihs.uni-stuttgart.de

Both models have in common that liquid and vapor phase share a common pressure field. For more detailed information about the two-phase modeling approaches it has to be referred to literature (e.g. [6, 7]).

The mass transfer between the phases is considered by the Zwart cavitation model [8], which contains model constants F_e and F_c for evaporation and condensation. Contrary to recommendations by Zwart et al. [8] to set F_e=50 and F_c=0.01, Morgut et al. [9] used the constants F_e=300 and F_c=0.03 that were determined by an optimization on a NACA66(mod) test case with sheet cavitation. In this study both suggested model constants are applied.

For RANS simulations, the SST turbulence model is chosen, which is widely used for CFD applications. As second model the SBES model [10] is selected, that can operate in scale resolving simulation (SRS) mode. The idea of SRS is to have a hybrid RANS-LES model that resolves large eddies away from the wall, while for the wall boundary layers a RANS model is used [11].

Computational setup

Extensive measurements have been performed for the NACA0009 hydrofoil in the high speed cavitation tunnel of the EPFL [2, 3, 4]. The hydrofoil has an initial chord of c_0=0.11 m, a maximum thickness h=9.9 mm and a span of 0.15 m. It is truncated to a chord of c=0.1 m and the foil tip on the pressure side of the blade is rounded with a radius of 1 mm to reduce the cavitation in the clearance [3]. This radius is considered in the simulation model, but to facilitate the meshing, a small radius of 0.1 mm remains on the suction side of the blade. Stereo PIV measurements of all velocity components in three different measurement planes have been carried out for non-cavitating conditions. The planes (see figure 1) are located downstream of the hydrofoil at z/c=1, 1.2 and 1.5.

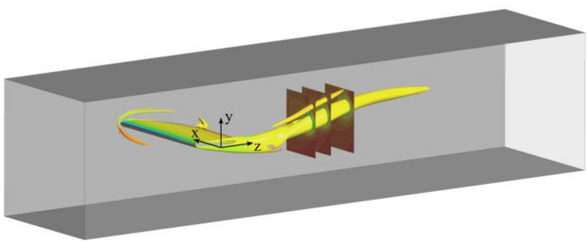


Figure 1(a): Measurement planes and visualization of the tip leakage vortex with an isosurface of the velocity invariant Q.

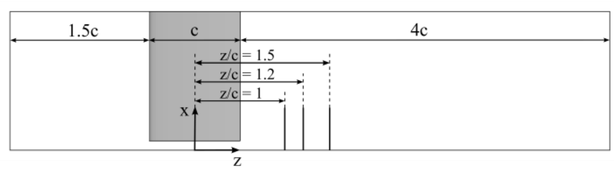


Figure 1(b): Top view on the main subdomain.

For all simulations a configuration is chosen with 10° incidence angle and a normalized tip clearance $\tau=\frac{gap}{h}=1$. As inlet boundary condition a block profile with a velocity w_∞=9.78 m/s and a turbulence intensity of 5% is set. This corresponds to the measurement with w_∞=10 m/s and takes into account the boundary layer close to the wall [3]. At the outlet, the static pressure is prescribed to obtain a pressure at the inlet p_∞=1 bar. To ensure that the boundary conditions do not affect the solution, the inlet boundary condition is set 3.5 chords upstream and the outlet boundary condition 12 chords downstream of the hydrofoil. The simulation domain is divided into three subdomains to reduce the number of cells. While the main subdomain (see figure 1(b)) around the hydrofoil has a fine resolution to resolve the tip leakage vortex, the other two subdomains consist of a coarser grid.

For all unsteady simulations, a second order backward Euler scheme is applied for temporal discretization and a time step of 10^{-5} s is prescribed, which corresponds to a RMS Courant number of 0.35. The selection of spatial discretization depends on the turbulence model. A high resolution scheme is prescribed for SST simulations and a bounded central differencing scheme is used for the SBES model.

Mesh study and turbulence model selection

A mesh study is carried out for the selection of a suitable grid. Steady state single phase simulations are performed using the SST turbulence model. In the first step, the global number of cells is varied, while the resolution of the boundary layer is kept constant, which results in an average y^+-value of 35. To quantify the differences between the grids, u-component of the velocity is compared to measurement results along a line that is chosen to pass the region of highest velocity gradient from the measurements. The results (see figure 2(a)) indicate that the smallest meshes 4M and 11M still show mesh dependencies, while the other grids have only minor differences in the velocity distribution. Nevertheless, there is still a huge deviation to the measurements that were performed at non-cavitating conditions.

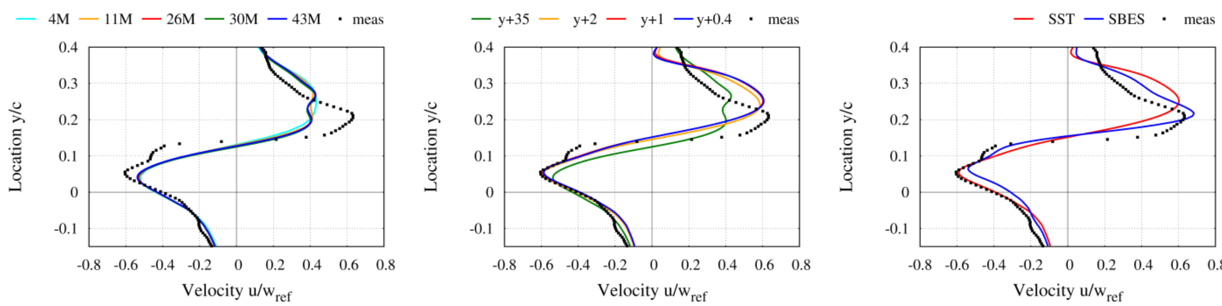
While for all other variants the mesh is globally refined, grid 30M is derived from variant 26M by locally refining the grid in the region of the tip leakage vortex. In simulations made on a Francis turbine with cavitating inter-blade

*Corresponding Author, Jonas Wack: jonas.wack@ihs.uni-stuttgart.de

vortices it was identified that mesh resolution in the vortex core region is of great importance to resolve the pressure minimum [12]. Thus, further investigations are based on grid 30M.

After the specification of the global cell distribution, meshes with different y^+-value are compared. In this step, only the resolution of the boundary layer of the wing and channel wall is varied, while the distribution in the rest of the domains remains constant. The velocity distribution of u is displayed in figure 2(b). It can be observed that the y^+-value has a significant impact on the simulation results. With decreasing y^+-value, simulation results approach to the experiment but the steep gradient from the measurements is still underestimated. At y^+=1, mesh independent results are observed, as a further refinement of the wall resolution has no relevant impact on the solution. Due to mesh independency and the best agreement with the measurements for the velocity distribution, the mesh with y^+=1 is selected for further investigations. Owing to the refinement in the boundary layers, this mesh consists in total of 46M cells.

Unsteady single-phase simulations are performed for the selection of the turbulence model. The results are displayed in figure 2(c). The SST simulation results are almost identical with the steady state simulations from the mesh study. With the more advanced SBES model, a significant improvement can be observed, which underlines that the strong swirling flow caused by the vortex needs a more complex turbulence modelling. Consequently, for the two-phase simulations the SBES model is applied.



(a) Variation of global mesh size. (b) Variation of mesh resolution. (c) Comparison of turbulence models.

Figure 2: Distribution of velocity u along line in y-direction at measurement plane z/c=1. The line is located at x/c=0.12.

Numerical results

The main focus in this study lies on the comparison of the homogeneous and inhomogeneous two-phase modeling approach. Two different configurations of cavitation model constants are prescribed for the homogeneous model. The simulations with the inhomogeneous model differ in the treatment of the dispersed vapor phase. Two simulations are performed with monodisperse settings, by prescribing a constant bubble diameter of the vapor phase of 0.01 mm and 0.1 mm, respectively. Furthermore, one simulation is performed with a polydispersed treatment based on the homogeneous MUSIG approach with ten size groups, a maximum bubble diameter of 1 mm and a minimum bubble diameter of 0.001 mm. The bubble diameters are selected based on the investigations by Maeda et al. [13] that state that the bubble radius in cavitation clouds is in the range between 0.001 and 0.1 mm.

A constant drag coefficient C_D=0.44 is prescribed for all the described simulations with the inhomogeneous model. Furthermore, one simulation is carried out with the Schiller Naumann drag model and a monodispersed vapor phase with 0.1 mm bubble diameter to investigate the impact of drag model. Other interfacial forces like lift force are neglected in this study.

In figure 3, a side view on the cavitation regions is displayed. On the left side an instantaneous snapshot is shown and on the right side the time-averaged results over 20,000 time steps are presented. The cavitation volume in the simulation results is displayed by isosurfaces with three different thresholds of the vapor volume fraction α_v: 0.5, 0.2 and 0.05. The use of three thresholds is intended to facilitate the assessment of cavitation occurrence. There are three different forms of cavitation present for the investigated NACA0009 hydrofoil. At the suction side of the leading edge, cloud cavitation can be observed due to the high incidence angle. Secondly, in the gap it comes to cavitation due to the formation of a tip separation vortex. Finally, a cavitating tip leakage vortex occurs.

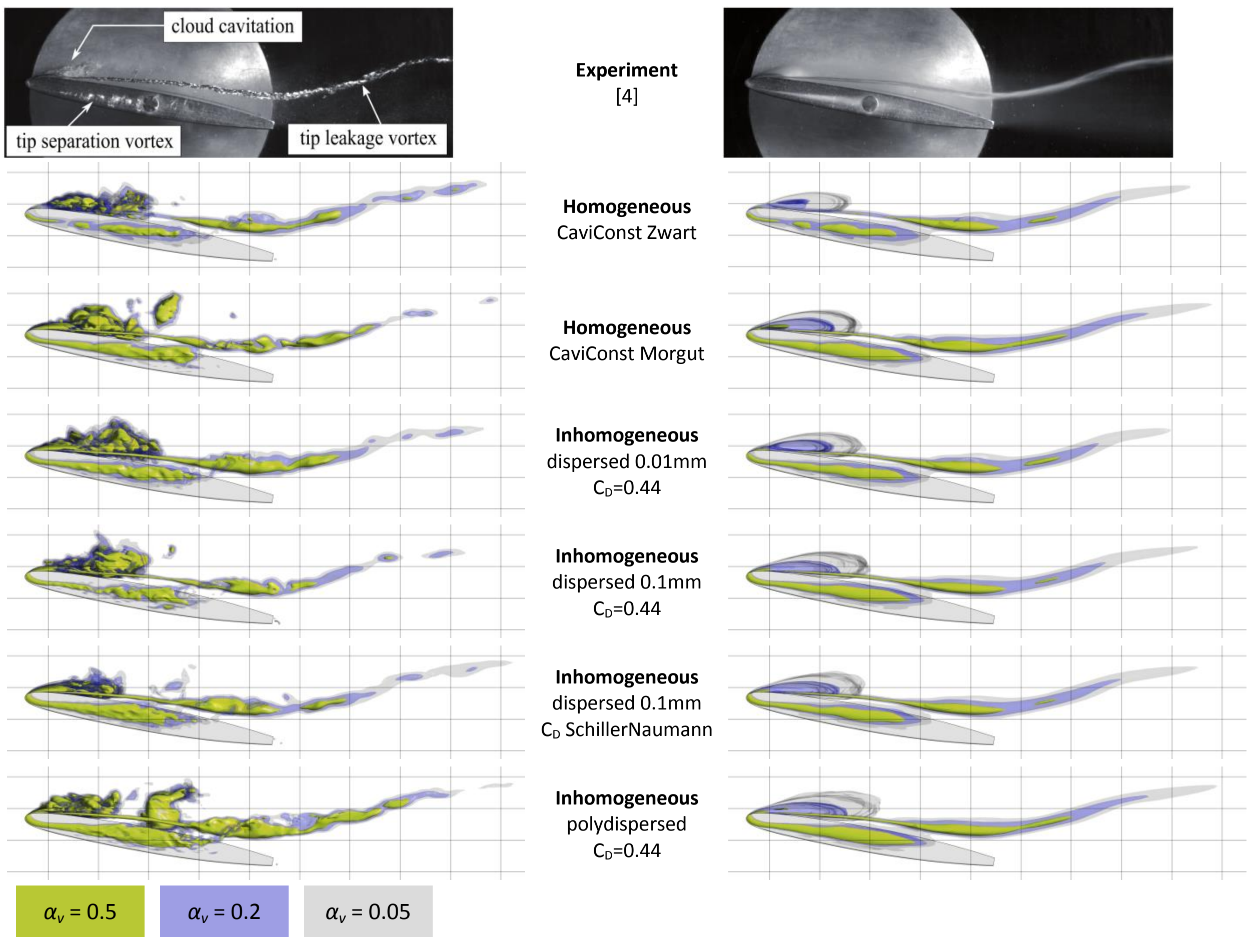
A comparison of the simulations with the homogeneous model indicates that for the averaged quantities, the cavitation volume is slightly bigger with the cavitation model constants from Morgut et al. [9]. The simulation results with the inhomogeneous model indicate a slight increase in vapor volume going from bubble diameter

*Corresponding Author, Jonas Wack: jonas.wack@ihs.uni-stuttgart.de

0.01 mm to 0.1 mm to a polydispersed treatment. The change in the drag model does not show any clear change in the results. A comparison between results from homogeneous and inhomogeneous model does not show any significant difference that would justify the additional computational effort that is caused by the inhomogeneous model. The similar results between the different two-phase modeling approaches can be explained by the Stokes number *St*:

$$St = \frac{\tau_V}{\tau_F} = \frac{\frac{\rho_v D^2}{18\mu_l}}{\tau_F}$$

The symbol τ_V denotes for the momentum response time and depends on the density of the vapor phase ρ_v, the bubble diameter D and the dynamic viscosity of the liquid phase μ_l. For St<<1, which corresponds to small bubble diameters, the response time of the vapor bubble is very fast compared to the characteristic time scale of the flow τ_F. The consequence is that the bubble has enough time to adapt to changes in the velocity field of the liquid phase, which results in a velocity equilibrium between the two phases [14]. As a result, the inhomogeneous model does not have any benefit in that case. On the other hand for big bubble diameters the two-fluid model might be beneficial, that is why it is often applied to bubble column simulations.



(a) Instantaneous snapshot. (b) Time-averaged result over 20,000 time steps.

Figure 3: Visualization of the cavitation volume by three isosurfaces of the vapor volume fraction with different thresholds.

All in all, it can be stated that all simulation results show a reasonable agreement with the measurement. Deviations can be found for the tip separation vortex, where the cavitation volume is overestimated in the region close to the leading edge, and in the region where the tip leakage vortex unites with the tip separation vortex. There, as well the cavitation volume is overestimated compared to the experiment. The deviations between the different simulation

*Corresponding Author, Jonas Wack: jonas.wack@ihs.uni-stuttgart.de

setups are relatively small. For this case it is more important to have a fine mesh resolution, especially to resolve the pressure minimum in the vortex core of the tip leakage vortex.

Conclusion

The selection of a suitable two-phase modeling approach is mandatory for accurate simulation results. For the presented investigations of the cavitating flow around a NACA0009 hydrofoil at 10° incidence angle, the inhomogeneous model does not show significant improvements compared to the simpler homogeneous model. The reason is that due to the small size of the vapor bubbles it comes to a strong coupling between the liquid and vapor phase, which results in velocity equilibrium.

For the selected case, it is more important to use fine grids that resolve the pressure minimum in the core of the tip leakage vortex. This is essential as the pressure level is linked with the mass transfer due to cavitation.

Acknowledgements

The authors would like to thank the HLRS Stuttgart for support and supply of computation time. Furthermore, the authors would like to thank the EPFL for providing the experimental data.

References

[1] Wack, J., Riedelbauch, S., Yamamoto, K., Avellan, F. (2016). *Two-phase flow simulations of the inter-blade vortices in a Francis turbine.* Proceedings of the 9th International Conference on Multiphase Flow, Florence, Italy.

[2] Dreyer, M., Decaix, J., Münch-Alligné, C., Farhat, M. (2014). *Mind the gap - tip leakage vortex in axial turbines*. IOP Conference Series: Earth and Environmental Science 22(5).

[3] Dreyer, M., Decaix, J., Münch-Alligné, C., Farhat, M. (2014). *Mind the gap: a new insight into the tip leakage vortex using stereo-PIV*. Experiments in Fluids. 55(11).

[4] Dreyer, M. (2015). *Mind The Gap: Tip Leakage Vortex Dynamics and Cavitation in Axial Turbines*. Ph.D. Thesis, EPFL Lausanne.

[5] Decaix, J., Balarac, G., Dreyer, M., Farhat, M., Münch, C. (2015). *RANS computations of tip vortex cavitation*. Journal of Physics: Conference Series. 656(1).

[6] Yeoh, G.H., Tu, J. (2010). *Computational Techniques for Multi-Phase Flows*. Elsevier.

[7] Ishii, M., Hibiki, T. (2010). *Thermo-Fluid Dynamics of Two-Phase Flow*. Springer.

[8] Zwart, P.J., Gerber, A.G., Belamri, T. (2004). *A Two-Phase Flow Model for Predicting Cavitation Dynamics*. Proceedings of the 5th International Conference on Multiphase Flow, Yokohama, Japan.

[9] Morgut, M., Nobile, E., Bilus, I. (2011). *Comparison of mass transfer models for the numerical prediction of sheet cavitation around a hydrofoil*. International Journal of Multiphase Flow. 37(6).

[10] Menter, F. (2016). *Stress-Blended Eddy Simulation (SBES) – A new Paradigm in hybrid RANS-LES Modeling*. Proceedings of the 6th HRLM Symposium, Strasbourg, France.

[11] Menter, F., Schütze, J., Gritskevich, M. (2012). *Global vs. Zonal Approaches in Hybrid RANS-LES Turbulence Modelling*. Progress in Hybrid RANS-LES Modeling. Springer.

[12] Wack, J., Riedelbauch, S. (2015). *Numerical simulations of the cavitation phenomena in a Francis turbine at deep part load conditions*. Journal of Physics: Conference Series. 656(1).

[13] Maeda, M., Yamaguchi, H., Kato, H. (1991). *Laser holography measurement of bubble population in cavitation cloud on a foil section*. Proceedings of the 1st ASME-JSME Fluids Engineering Conference, Portland, USA.

[14] Crowe, C.T., Schwarzkopf, J.D., Sommerfeld, M., Tsuji, Y. (2011). *Multiphase Flows with Droplets and Particles*. CRC Press.

*Corresponding Author, Jonas Wack: jonas.wack@ihs.uni-stuttgart.de