Application of a compressible flow solver and barotropic cavitation model for the evaluation of the suction head in a low specific speed centrifugal pump impeller channel

P Limbach, T Müller and R Skoda
Chair of Hydraulic Fluid Machinery, Ruhr-Universität Bochum, Universitätsstr. 150, 44801 Bochum, Germany
E-mail: phillip.limbach@ruhr-uni-bochum.de

Abstract. Commonly, for the simulation of cavitation in centrifugal pumps incompressible flow solvers with VOF kind cavitation models are applied. Since the source/sink terms of the void fraction transport equation are based on simplified bubble dynamics, empirical parameters may need to be adjusted to the particular pump operating point. In the present study a barotropic cavitation model, which is based solely on thermodynamic fluid properties and does not include any empirical parameters, is applied on a single flow channel of a pump impeller in combination with a time-explicit viscous compressible flow solver. The suction head curves (head drop) are compared to the results of an incompressible implicit standard industrial CFD tool and are predicted qualitatively correct by the barotropic model.

1. Introduction
For the complex cavitating flow in centrifugal pumps, commonly, incompressible flow solvers with an additional transport equation for the void fraction are used (e.g. [1]). Mass transfer is calculated by a source/sink term based on a simplified bubble dynamics equation, such as the Rayleigh equation (e.g. [2, 3]). Due to the simplification of the physics, empirical parameters are implemented in the source/sink terms, which may need to be calibrated to the particular flow situation. In contrast, a barotropic equation of state is used by several authors (i.e. [4, 5, 6]). While [4] use an isothermal barotropic model function for the mixture and [5] use a barotropic relation, which accounts for effects of thermal cavitation, the model of [6] is based on an isentropic path in the temperature-entropy diagram. This model have been applied e.g. on micro throttles [7], on hydrofoils [8] (companion paper) and ultrasonic horn flows [9], [10] (companion paper), because of their accurate resolution of collapse-induced wave dynamics, which lead to erosion damages. Moreover, the structure and measured shedding frequency of cloud cavitation can be captured with a good accuracy although the grid sensitivity and computational effort are high [8]. In the present study the barotropic cavitation model of [6] is applied on a single flow channel of a centrifugal pump with low specific speed, in order to evaluate the potential of this model to predict the pump head in cavitating flow conditions. The investigated pump is a single-stage glanded centrifugal pump of low specific speed ($n_q = 12 \text{ min}^{-1}$), which contains a closed impeller with seven blades and a volute. The geometry and operating conditions of the pump are documented in [11].
2. Method

2.1. Numerical and physical model

On the one hand the commercial flow solver Ansys CFX 15.0 is used to solve the incompressible Reynolds-averaged Navier-Stokes equations. Second order discretisation is used for time and space and the SST turbulence model [12] in combination with an automatic wall-function is applied. The two-phase flow is treated as a homogeneous mixture and described by an additional transport equation for the void fraction. The mass transfer model by Zwart et al. [3] is used and based on a simplified Rayleigh-Plesset equation [13, 14], i.e. Rayleigh equation, which governs the dynamics of a single, spherical bubble, a mass transfer term is included in the transport equation of the void fraction. For the present study parameters of the cavitation model are defined ($R_B = 10^{-6}$ m, $\alpha_{nuc} = 5e^{-4}$, $F_{vap} = 50$, $F_{cond} = 0.01$) as recommended by Zwart et al. [3]. This method is referred to as incompressible in the following.

On the other hand, our block-structured inhouse-code hydRUB contains a compressible flow solver with an explicit optimised four-stage Runge-Kutta time integration scheme [15]. The explicit time integration leads to very small time steps, hence, Reynolds-averaging of the compressible Navier-Stokes equations is not valid. As a consequence, viscous fluid is used but no explicit turbulence model is applied. The convective fluxes are modelled using a low Mach-number consistent Godunov-type flux formulation [15]. Second order discretisation in space is achieved by the reconstruction of density and velocity with the high resolution schemes MINMOD [16] and SMART [17], respectively. The fluid is treated as a homogeneous mixture of vapour and liquid assuming both phases to be in thermodynamic equilibrium. The energy equation is neglected and a barotropic cavitation model, i.e. equation of state is used [6] that assumes an isentropic phase change. No model parameters are available in contrast to the mass transfer model [3]. This method is referred to as compressible in the following.

2.2. Numerical set-up

In the present first application of the compressible method to the flow in a low specific speed centrifugal pump impeller, we assess the ability of the compressible method to predict the onset of head drop if the pressure level is successively decreased. A simplified geometry is used, i.e. a single flow channel of the impeller ($\approx 51^\circ$ segment). The volute and sidewall gaps are neglected. The suction pipe and the outlet of the impeller channel are extended to reduce the influence of boundary conditions on internal flow. However, the reflection of pressure waves particular at the inlet is unavoidable for the compressible flow solver by our reflecting boundary conditions. Nevertheless, we cope with the pressure wave reflections at the present first study, since we are interested in time averaged head values. A confuser is added at the radial outlet section to avoid positive pressure gradients in the outlet region. A relatively coarse grid is used (60,000 nodes). A preliminary grid study showed (120,000; 240,000 nodes) that for these grids the time-averaged head varies by less than 2% for single phase (both solvers) and cavitating flow conditions (only incompressible tested). A schematic view of the numerical set-up is shown in figure 1.

![Figure 1.](image-url)

Figure 1. Numerical set-up of the impeller in conformal mapping and numerical model in a schematic meridian view.

A constant velocity is specified at the inlet of the suction pipe corresponding to the best
efficiency point and for part- (−30%) and overload (+30%). Static pressure is set at the outlet, periodic boundary conditions are used in circumferential direction. Instationary simulations are performed in relative frame with frozen rotor. The convergence criterion is defined as \( R_{\text{max}} \leq 1 \times 10^{-4} \) and the time step size is set to a value corresponding to a rotation of 1° of the impeller for the incompressible solver. The time-explicit scheme of the compressible solver leads to time steps of approximately \( 2.8 \times 10^{-8} \)s \((CFL \leq 0.95)\). Concerning single phase flow simulations the measured pump head is overpredicted even with a full model containing a 360° geometry of the impeller, sidewall gaps and the volute [11]. The single channel flow model contains more geometry simplifications, hence, we do not focus on achieving quantitative agreement between simulation results and measured data, but rather on a first assessment of the barotropic model to capture the head drop curve.

3. Results

The pressure level at the outlet is defined high enough to ensure non-cavitating flow conditions (3 bar). For the prediction of the characteristics of the Net Positive Suction Head (NPSH) the pressure level is decreased successively. As a consequence, cavitation zones appear and grow, leading to a head drop. The computed head over time is presented exemplarily in figure 2.

For single phase flow conditions a constant head is predicted using the incompressible flow solver. In contrast, oscillations occur in the results of the compressible flow solver, which correlate with the length of the flow path and the speed of sound \((\sim 1480 \text{ m/s})\) since pressure waves are reflected at the inlet and outlet boundaries. Regarding cavitating flow conditions only minor fluctuations are visible for the incompressible flow solver. In the compressible solution the void regions oscillate with a relatively low frequency and lead to corresponding head oscillations. Occasionally occurring vapour region collapses close to the leading edge or suction pipe section generate pressure waves, which are propagated to the boundaries and lead to a sudden head drop (e.g. figure 2, 0.06s). Although the occurrence of cloud cavitation is likely for low NPSH [19], only sheet cavitation is observed in our simulation results, which may be attributed to the rather coarse grid.

Figure 3 and 4 presents the results of the head for single phase and cavitating flow conditions, respectively, which is averaged over a sufficiently long time interval. Both solvers predict essentially the same head characteristics for single phase flow conditions, while slight deviations in the overload operating point can be observed. Due to only small head deviations between both methods, the influence of the turbulence modelling (eddy viscosity versus no explicit turbulence model) seems to be minor for the evaluation of the head: Regarding the compressible solver we assume that the numerical diffusion acts as a subgrid-scale eddy viscosity. We expect that the local flow field in cavitating flow conditions is certainly influenced by a grid-induced arbitrary subgrid-scale viscosity magnitude, but time averaged head values do not seem to be dominated by turbulence effects, a conclusion which is also supported by [18]. In comparison to [11] an essentially linear slope of the head curve and higher deviation between simulation results and measured data (not shown here) is obtained due to the neglect of the volute and side chambers.

The head drop due to blockage of the flow channel with vapour can be captured by both methods. For all operating points, a sharp drop of the head is predicted. In nominal load
A local increase of the head is observed ($NPSH \sim 0.5 \text{ m}$) using the compressible flow solver. Concerning this operating point, the flow around the blade leading edge is influenced by the vapour zones and attached flow on the pressure side is achieved downstream, whereas backflow is observed in the corresponding results of the incompressible flow solver. This qualitatively different result of both solvers needs to be verified on finer grids. Differences in the head already present in figure 3 (overload) are also present at lower NPSH values (figure 4, overload). The resulting $NPSH_{3\%}$ (NPSH at a head drop of three percent) increases with increasing flow rate as expected from common theoretical and experimental findings (e.g. [19]).

4. Conclusions
The suction head curves (head drop) are predicted qualitatively correct by the barotropic model, although no empirical model parameters have been adjusted. This promising performance needs to be verified on finer grids and more realistic pump models including sidewall gaps and volute casing, which is done in ongoing work by the authors.

Acknowledgments
The authors would like to express their gratitude to the Verband Deutscher Maschinen- und Anlagenbau (VDMA) for financial support. Further, the authors are grateful to University of Cologne for the grant of computing time on the Cheops Compute Cluster and gratefully acknowledge the company WILO SE for the supply of the pump data.

References
[1] Iiu D, Iiu H, Wang Y, Zhuang S, Wang J and Hui D 2012 Proc. 8th Int. Symp. on Cavitation (Singapore)
[2] Schnerr G H and Sauer J 2001 Proc. 4th Int. Conf. on Multiphase Flow (New Orleans)
[3] Zwart P J, Gerber A G and Belamri T 2004 Proc. 5th Int. Conf. on Multiphase Flow (Yokohama)
[4] Goncalves E and Patella R F 2009 Computers and Fluids 38 (9) pp 1682-96
[5] Rapposelli E and d’Agostino L 2003 Proc. 5th Int. Symp. on Cavitation (Osaka)
[6] Iben U 2002 Systems Analysis Modelling Simulation 42 pp 1283-307
[7] Skoda R, Iben U, Morozov A, Mihatsch M, Schmidt S J and Adams N A 2011 WIMRC Proc. 3rd Int. Cavitation Forum (Warwick)
[8] Bühme M and Skoda R 2015 Proc. 9th Int. Symp. on Cavitation (Lausanne)
[9] Mottyll S, Mueller S, Niederhofer P, Hussong J, Huth S and Skoda R 2013 EPJ Web of Conferences 67
[10] Mottyll S and Skoda R 2015 Proc. 9th Int. Symp. on Cavitation (Lausanne)
[11] Limbach P, Kimoto M, Deimel C and Skoda R 2014 Proc. of ASME Turbo Expo (Düsseldorf)
[12] Menter F R 1994 AIAA Journal 32 pp 1598-605
[13] Rayleigh L 1917 Phil. Mag. 34 pp 94-8
[14] Plesset M S 1949 ASME J. of Appl. Mech. 16 pp 228-31
[15] Schnerr G H, Sezal I H and Schmidt S J 2008 Physics of Fluids 20 040703
[16] Harten A 1983 J. Computational Physics 49 pp 357-93
[17] Gaskell P H and Lau A K C 1988 Int. J. Numerical Methods in Fluids 8 pp 617-41
[18] Wursthorn S and Schnerr G H 2002 Proc. in Applied Mathematics and Mechanics 1 pp 375-6
[19] Gülich J F 2014 Centrifugal pumps (Springer, Berlin Heidelberg)