Numerical and experimental investigation of the cavitating flow in a low specific speed centrifugal pump and assessment of the influence of surface roughness on head prediction

Phillip Limbach†*, Tim Müller†, Martin Blume†, Romuald Skoda†

Abstract
In order to assess the numerical model for the prediction of the pump performance of a low specific speed centrifugal pump in cavitating flow conditions a numerical 3D analysis with a state-of-the-art CFD method and cavitation model by Zwart et al. [1] is performed for both, design and off-design conditions with varying surface roughness. Measured head and suction head data of in-house experiments with two different volute roughness levels are used to validate the unsteady simulation results. Regarding single-phase flow, significant differences to the measured head are observed assuming hydraulically smooth walls in the numerical model, in particular for over load. By employing a surface roughness model the simulation results show a decrease of the head at higher flow rates according to the measurements. Further deviations between simulation and measured data at over load are attributed to flow separation at the tongue of the volute, which is inaccurately resolved by the wall function approach. For cavitating flow conditions the influence of the surface roughness on the 3% suction head is minor. The trend of the measured $NPSH_{3\%}$ curve is qualitatively reproduced by the simulation. A steep rise of the $NPSH_{3\%}$ curve towards over load is attributed to cavitation at the volute tongue.

1 Chair of Hydraulic Fluid Machinery, Ruhr-Universität Bochum, Germany
*Corresponding author: phillip.limbach@ruhr-uni-bochum.de

INTRODUCTION
Centrifugal pumps are increasingly required to operate in an extended range of flow rates, which deviate significantly from the design point, leading to unsteady flow instabilities, performance losses and cavitation. Cavitation may lead to unstable pump operation, noise, head drop and even material erosion. Despite the fact that the operation disturbance due to cavitation is well known, it is often not taken into account in common engineering CFD simulations because of the complexity of the cavitation physics, e.g. unsteadiness, bubble dynamics, evaporation and condensation or air release.

Models for the simulation of cavitating flows are principally divided into interface-tracking, Volume of Fluid (VoF), discrete-bubble as well as two-phase flow models [2]. In fact, no cavitation model resolves the physics of all above mentioned phenomena. For engineering simulations of centrifugal pumps, incompressible, implicit flow solvers with VoF-methods and a transport equation for the void fraction are usually employed due to their moderate computational effort and good solver stability. Mass transfer between the phases is calculated by a source/sink term based on a simplified bubble dynamics equation, such as the Rayleigh equation (e.g. [1, 3]). In order to compensate the physical simplifications of the Rayleigh equation empirical parameters are implemented in the source/sink terms, which may need to be calibrated to the particular flow situation. This model class has been applied for the simulation of cavitating flow in centrifugal pumps of high or intermediate specific speed in order to evaluate the characteristics of head drop and net positive suction head ($NPSH$), and the measured $NPSH_{3\%}$ curve has been reproduced with reasonable accuracy by the simulations (e.g. [4, 5, 6]). However, the accuracy of the prediction of suction performance is case-dependent according to Nohmi [7], in particular for low specific speed pumps.

Simulation results of low specific speed centrifugal pumps commonly deviate from measured performance data considerably, even for single-phase flow conditions [8, 9, 10, 11, 12]. Due to narrow flow channels the relative surface roughness increases with decreasing nominal pump size. As a result, the performance of the pump is influenced significantly by the surface finish [13]. Varley [14], Tamm et al. [15] and Güllich [16] varied the roughness of various impeller components of different centrifugal pumps and registered only a slight increase of the head with increasing roughness in their experiments. Tamm et al. [15] and Güllich [16] identified the roughness of the volute to have a major influence on the head, which decreases significantly with a roughened volute. Juckelant et al. [11] figured out that the neglect of surface roughness in the numerical model causes significant differences between simulation results and measured data of low specific speed
Numerical and experimental investigation of the cavitating flow in a low specific speed centrifugal pump and assessment of the influence of surface roughness on head prediction — 2/9

1. Tank
2. Closed ball valves
3. Piezoresistive pressure transducer (on the suction side)
4. Piezoresistive pressure transducer (on the pressure side)
5. Magnetic inductive flow meter
6. Throttle
7. Connection to the vacuum pump
8. Vacuum pump

CP1 Centrifugal pump 1 (\(n_q = 12 \text{ min}^{-1}\))
CP2 Centrifugal pump 2 (not used here)

Figure 1. Closed circuit centrifugal pump test rig.

centrifugal pumps. In addition, Juckelant and Wurm [12] used a low specific speed centrifugal pump with polished surfaces and reveal that flow separation at the tongue of the volute is inaccurately predicted by the wall function approach, hence, a high resolution of the boundary layer is necessary for a reliable prediction of the flow field and pump performance.

The present paper focuses on the assessment of the numerical model for the prediction of the head drop due to cavitation in a low specific speed centrifugal pump. In-house experiments are carried out in order to validate the simulation results. The commercial 3D CFD code ANSYS CFX 15.0 is employed. For single-phase flow, the influence of several geometry and model simplifications as well as of the grid resolution on the pump performance is analysed. For single-phase as well as cavitating flow, the surface roughness is varied in the simulation as well as experiment.

1. METHODOLOGY

1.1 Experimental set-up

Experiments are performed at the closed circuit centrifugal pump test rig of the Chair of Hydraulic Fluid Machinery at Ruhr-Universität Bochum. The test rig is equipped with piezoresistive pressure transducers to measure the static pressure up- and downstream of the pump as well as a magnetic inductive flow meter and a throttle to adapt the volume flow. A stroboscope is used to verify a constant rotational speed. In order to reduce the impact of flow inhomogeneity up- and downstream of the pump the undisturbed suction and pressure pipes extend 32\(D\) and 47\(D\), respectively, \(D\) being the pipe diameter. Utilising a vacuum pump in the tank of the test rig, the absolute system pressure is variable between 1 bar to 0.05 bar within about 10 minutes. Measured data are recorded every \(\Delta t = 0.5 \text{s}\) until the \(NPSH_{3\%}\) is determined at 3% head drop. This procedure is repeated five times per flow rate and the standard deviation of the \(NPSH_{3\%}\) is determined and depicted as error bars in figure 6. In comparison to exemplary stationary measurements at a multitude of distinct tank pressure levels, i.e. the common \(NPSH_{3\%}\) measurement procedure, a maximum deviation regarding the \(NPSH_{3\%}\) of 10% is observed at over load, which is considered to be acceptable in the present study. The test rig is shown in figure 1. The ball valves to CP2 are closed to prevent a bypass flow in the experiments.

Table 1. Pump performance data.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nominal rotational speed</td>
<td>1450 min(^{-1})</td>
</tr>
<tr>
<td>Nominal flow rate</td>
<td>15 m(^3)/h</td>
</tr>
<tr>
<td>Nominal head</td>
<td>15.5 m</td>
</tr>
<tr>
<td>Nominal power consumption</td>
<td>1.5 kW</td>
</tr>
<tr>
<td>Specific speed</td>
<td>12 min(^{-1})</td>
</tr>
<tr>
<td>Flange nominal diameter on the suction side</td>
<td>DN 50</td>
</tr>
<tr>
<td>Flange nominal diameter on the pressure side</td>
<td>DN 32</td>
</tr>
</tbody>
</table>

A single-stage low specific speed (\(n_q = 12 \text{ min}^{-1}\)) centrifugal pump (CP1 in figure 1) with a seven blade closed impeller and spiral volute is investigated. The nominal rotational speed and nominal flow rate at the best efficiency point equal \(n = 1450 \text{ min}^{-1}\) and \(Q_{opt} = 15 \text{ m}^3/\text{h}\), respectively. Since in low specific speed centrifugal pumps the volute has a major influence on the pressure losses [15, 16], two different surface qualities are investigated for the volute. On the one hand the untreated cast iron surface is kept, on the other hand the roughness is decreased by cathodic dip
A mass source term in the vapour volume fraction equation accounts for the mass exchange between the phases, liquid and vapour. The ANSYS CFX 15.0 implementation of the cavitation model accounts for the mass exchange between the phases, liquid and vapour, as well as transport equations of the turbulence model. The volume fraction $\alpha$ to the constraint $\text{Re}$ is subject to the constraint $\alpha_l + \alpha_v = 1$. The mass fraction equation 1 is reformulated as a volume fraction equation. The governing equation set consists of the mixture continuity equation, mixture momentum equations, the vapour volume fraction equation as well as transport equations of the turbulence model. A mass source term in the vapour volume fraction equation accounts for the mass exchange between the phases, liquid water and vapour ($\dot{S}_{\text{vap}} = -\dot{S}_{\text{cond}}$) and is calculated on the basis of the Rayleigh equation (equation 3), which relates the temporal change of the bubble radius to the difference between ambient static pressure and vapour pressure [17].

$$\frac{dR}{dt} = \sqrt{\frac{2}{3}} \frac{p_v - p}{p_l}$$  

Inserting the Rayleigh equation in the rate of change equation of mass of a single bubble and specifying $N_B = \frac{\alpha_v}{\frac{4}{3} \pi R^3}$, the total interphase mass transfer rate per unit volume is formulated in equation 4 for vapourisation ($p < p_v$) and condensation ($p > p_v$). Empirical coefficients, $F_{\text{vap}}$, $F_{\text{cond}}$, $R_B$ and $\alpha_{\text{nuc}}$, are employed to compensate the simplifications in the Rayleigh equation and to account for different time scales of vapourisation and condensation.

$$\dot{S}_{\text{vap, cond}} = \begin{cases} F_{\text{vap}} \frac{3\alpha_{\text{nuc}}(1-\alpha_v)p_v}{R_B} \sqrt{\frac{2}{3}} \frac{p_v - p}{p_l} & \text{if } p < p_v \\ F_{\text{cond}} \frac{3\alpha_{\text{nuc}}p_v}{R_B} \sqrt{\frac{2}{3}} \frac{p - p_v}{p_l} & \text{if } p > p_v \end{cases}$$  

Due to the assumption of constant temperature ($T = 298.15$ K), the density of the liquid, $\rho_l$, and vapour, $\rho_v$, as well as the vapour pressure, $p_v$, are assumed to be constant (cf. table 3). The empirical coefficients, $F_{\text{vap}}$, $F_{\text{cond}}$, $R_B$ and $\alpha_{\text{nuc}}$, are specified as recommended by Zwart et al. [1].

### 1.2 Cavitation model

The ANSYS CFX 15.0 implementation of the cavitation model of Zwart et al. [1] is utilised and briefly summarised in the following. Assuming the two-phase flow as a homogeneous mixture of incompressible phases, the governing equations of the two-phase flow are based on the continuity equation for each phase and conservation equations for momentum of the mixture, cf. equation 1 and 2. Thermal effects are neglected, thus the energy equation is not solved. This is equivalent with the assumption of an isothermal flow. The turbulent eddy viscosity of the mixture, $\mu_{tm}$, appears in the Reynolds averaged Navier-Stokes equations.

$$\frac{\partial (\alpha_k \rho_k)}{\partial t} + \frac{\partial (\alpha_k \rho_k u_i)}{\partial x_i} = \dot{S}_k$$  

$$\frac{\partial (\rho_m u_i)}{\partial t} + \frac{\partial (\rho_m u_j u_i)}{\partial x_j} =$$

$$- \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ (\mu_m + \mu_{tm}) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right]$$

The volume fraction $\alpha_k$ of both phases ($k = l, v$) is subject to the constraint $\alpha_l + \alpha_v = 1$. The mass fraction equation 1 is reformulated as a volume fraction equation. The governing equation set consists of the mixture continuity equation, mixture momentum equations, the vapour volume fraction equation as well as transport equations of the turbulence model. A mass source term in the vapour volume fraction equation accounts for the mass exchange between the phases, liquid water and vapour ($\dot{S}_{\text{vap}} = -\dot{S}_{\text{cond}}$) and is calculated on the basis of the Rayleigh equation (equation 3), which relates the temporal change of the bubble radius to the difference between ambient static pressure and vapour pressure [17].

$$\frac{dR}{dt} = \sqrt{\frac{2}{3}} \frac{p_v - p}{p_l}$$  

Inserting the Rayleigh equation in the rate of change equation of mass of a single bubble and specifying $N_B = \frac{\alpha_v}{\frac{4}{3} \pi R^3}$, the total interphase mass transfer rate per unit volume is formulated in equation 4 for vapourisation ($p < p_v$) and condensation ($p > p_v$). Empirical coefficients, $F_{\text{vap}}$, $F_{\text{cond}}$, $R_B$ and $\alpha_{\text{nuc}}$, are employed to compensate the simplifications in the Rayleigh equation and to account for different time scales of vapourisation and condensation.

$$\dot{S}_{\text{vap, cond}} = \begin{cases} F_{\text{vap}} \frac{3\alpha_{\text{nuc}}(1-\alpha_v)p_v}{R_B} \sqrt{\frac{2}{3}} \frac{p_v - p}{p_l} & \text{if } p < p_v \\ F_{\text{cond}} \frac{3\alpha_{\text{nuc}}p_v}{R_B} \sqrt{\frac{2}{3}} \frac{p - p_v}{p_l} & \text{if } p > p_v \end{cases}$$  

Due to the assumption of constant temperature ($T = 298.15$ K), the density of the liquid, $\rho_l$, and vapour, $\rho_v$, as well as the vapour pressure, $p_v$, are assumed to be constant (cf. table 3). The empirical coefficients, $F_{\text{vap}}$, $F_{\text{cond}}$, $R_B$ and $\alpha_{\text{nuc}}$, are specified as recommended by Zwart et al. [1].

### 1.3 Numerical set-up

The geometry model contains the seven blade impeller, volute casing, side gaps as well as the suction and pressure pipes, which are extended (>4D) in axial direction to reduce the effect of the boundary conditions on the internal flow. The computational domain and the grid of the impeller are shown in figure 2. Preliminary single-phase flow simulations have shown that the influence of geometry details, i.e. the radii between the blades and hub/shroud, ribs in the suction pipe, axial thrust balance holes as well as tolerances of the radial clearance of the gap between impeller shroud and casing on the predicted head is minor ($\Delta H < 1\%$). Hence, these geometry details are neglected in the numerical model.

A block structured grid is generated with three different resolutions containing approximately 1,850,000 (CWF), 16,330,000 (FWF) and 16,510,000 nodes (FLRe). The grids of the impeller, side gaps and suction pipe are generated with an in-house turbomachinery grid generator. The volute as well as the pressure pipe grids are generated with the commercial program ICEM CFD. Regarding the grid FWF, the length of the cell edges of CWF is at least halved in each direction. For the generation of the grid FLRe, additional cells are added in the boundary layer region of the volute of FWF in order to resolve the boundary layer flow down to a non-dimensional wall distance $y^+ \sim 1$. For the approximation of the Reynolds stresses the shear stress transport (SST) turbulence model [18] is used in combination with an automatic wall function. The wall treatment, i.e. the integration to the wall (low-Reynolds number model) or the matching of a logarithmic velocity profile (wall function), is blended automatically depending on the resolution of the grid near the wall [19, 20]. The wall adjacent resolution of the grids CWF and FWF is optimised for the use of wall functions, whereas the grid FLRe is constructed for a low-Reynolds number approach in the volute. The wall $y^+$ distribution is summarised in table 2.

According to Lechner and Menter [21], the surface roughness, which is described by an equivalent sand grain roughness of an average roughness height, is modelled by placing the wall nodes at 50% of the sand grain roughness height in combination with a modification of the automatic wall function,
which is shifted downwards depending on the dimensionless sand grain roughness height. The viscous sublayer is lost quickly with increasing dimensionless sand grain roughness height, hence, the blending function activates the viscous sublayer formulation only for small values of the dimensionless sand grain roughness height [21]. The relative error of the wall shear stress of an incompressible plane Couette flow increases with increasing grid resolution up to \(\sim 10\% \) [21].

Table 2. Average wall \(y^+\) for different grids and domains.

<table>
<thead>
<tr>
<th></th>
<th>Suction pipe</th>
<th>Pressure pipe</th>
<th>Impeller gaps</th>
<th>Volute</th>
</tr>
</thead>
<tbody>
<tr>
<td>CWF</td>
<td>60</td>
<td>100</td>
<td>60</td>
<td>100</td>
</tr>
<tr>
<td>FWF</td>
<td>30</td>
<td>50</td>
<td>30</td>
<td>50</td>
</tr>
<tr>
<td>FLRe</td>
<td>30</td>
<td>50</td>
<td>30</td>
<td>50</td>
</tr>
</tbody>
</table>

The equivalent sand grain roughness can be assessed on the basis of the measured arithmetic mean roughness \(R_a\) by using an equivalence coefficient, \(c_{eq}\) \((k_s = 6R_a/c_{eq})\) [13]. A wide range of equivalence coefficients that connect the arithmetical mean roughness and equivalent sand grain roughness is used in the literature (e.g. [15, 16, 13]). In order to circumvent this ambiguity, in the present investigation the equivalent sand grain roughness height is varied until the simulation results with the grid CWF match the measured data EXP_R39/8 at nominal load. An equivalent sand grain roughness height of \(k_s = 140\) \(\mu m\) is obtained, which corresponds to a rather small conversion coefficient and a high value of the equivalent sand grain roughness compared to the result of measured mean roughness, \(R_z \sim 39 \mu m\), and arithmetic mean roughness \(R_a \sim 8 \mu m\). Since Tamm et al. [15] and Gülich [16] have shown that the impeller roughness has a minor effect on the head, a fact which has been confirmed in preliminary simulations in the present study, the entire computational model (impeller, side gaps and volute) is equipped either with a hydraulically smooth (SMOOTH) or a rough wall \((k_s = 140)\). The successive case names of the simulation runs are composed of the grid resolution and the surface treatment.

A constant velocity is specified at the inlet of the suction pipe (turbulence intensity 5%) and a uniform static pressure is defined at the outlet of the pressure pipe. Unsteady single-phase and cavitating flow simulations are performed for five and three flow rates, respectively. A time step size corresponding to an impeller rotation of 1° is used. Second order discretisation in space and time as well as double precision floating point number accuracy is applied. Further details of the numerical set-up are listed in table 3. The performance quantities of the pump, which are described in section 2.1, are evaluated by means of sufficiently long time-averaging, at least over the complete final revolution. The approximate outlet pressure conditions for the determination of the \(\sim 3\%\) head drop and \(NPSH_{3\%}\) are available from steady state simulations, which have been performed in a previous study [10]. Due to the fact that a head drop of 3% is not matched precisely in the simulations, intermediate \(NPSH_{3\%}\) values are determined by means of linear interpolation.

2. RESULTS

2.1 Evaluation of performance quantities

The head is computed according to equation 5. Regarding the simulation results, the total pressure is mass-averaged at the evaluation surfaces ES1 and ES2 (cf. figure 2) considering a constant geodetic height difference of the evaluation surfaces of 0.24 m. In the experiments, the total pressure is determined by measuring the static pressure (pressure transducers), dynamic pressure (flow meter and pipe diameter) and the height
difference of the pressure transducers.

\[
H = \frac{p_{t,ES2} - p_{t,ES1}}{\rho g}
\]  

(5)

The net positive suction head \((NPSH)\) is evaluated in terms of the required \(NPSH\) of the pump \((NPSHR)\) which can be considered as a measure for the margin against vaporisation of the fluid in the pump.

\[
NPSH = \frac{p_{t,ES1} - p_v}{\rho g}
\]  

(6)

\(NPSH_{5\%}\) is evaluated at a 3\% head drop.

### 2.2 Single-phase flow

In a previous study \([10]\) the measured head is significantly overestimated by the simulation results with a computational grid that approximately corresponds to the resolution employed in the grid CWF of the present study. The result of Limbach et al. \([10]\) is essentially reproduced in the present study with hydraulically smooth walls and the grid CWF, cf. case SIM_CWF_SMOOTH in figure 3. Comparing the results of SIM_CWF_SMOOTH and SIM_FLRe_SMOOTH, as depicted in figure 3, no significant dependence of the grid resolution on the head can be observed for flow rates up to the nominal operating point \((Q_{opt} = 15 \text{ m}^3/\text{h})\). However, the resolution of the boundary layer of the volute in the SIM_FLRe_SMOOTH case leads to a noticeable decrease of head in over load operation. Regarding over load, the examination of the velocity profiles shows a high angle of incidence at the volute tongue, which varies periodically because of the movement of the blades relative to the volute tongue. As a result, flow separation occurs, which cannot be resolved by the wall function. Using the grid FWF (cf. section 1.3) no significant discrepancies to SIM_CWF_SMOOTH are observed (not shown here). Thus, the accurate resolution of the separation at the volute tongue is not a feature of the overall number of nodes but of the accurate resolution of the boundary layer \((y^+ \sim 1)\) with a low-Reynolds number wall approach. The low accuracy of the wall function due to its inability to predict flow separation in a low specific speed pump volute has been already pointed out by Juckelandt and Wurm \([12]\) and is thus confirmed in the present study.

![Figure 3. Head characteristics dependent on the grid resolution (hydraulic smooth surfaces in the simulation).](image)

In order to exclude the assumption of Reynolds stress isotropy in the eddy viscosity turbulence model as a further source of prediction uncertainty, a test on the grid CWF with the ANSYS CFX 15.0 implementation of the BSL Reynolds stress model, which accounts for the turbulence anisotropy, has been performed for over load and does not lead to significant changes of the head \((\Delta H < 1\%)\). Hence, the assumption of isotropic turbulence due to the scalar eddy viscosity of the SST turbulence model seems to be acceptable for the prediction of the head.

The influence of surface roughness on the head is illustrated in figure 4 and discussed in the following. Experiments with varying roughness of the surfaces show a trend of decreasing head with increasing surface roughness. Roughening of the surface leads to an increase of the friction losses and hence to a reduction of head at a fixed flow rate at over load. Since friction losses also depend on the Reynolds number, the head loss is more distinctive with increasing flow rate. The experimental data clearly figure this out. Simulation results accounting for surface roughness are presented for the grid CWF with an equivalent sand grain roughness...
height of $k_s = 140 \mu m$ (cf. section 1.3) and compared to the EXP_R39/8 measurement. Following from the adaption procedure of $k_s$, the simulated head is per definition equal to the measured head at nominal flow rate $Q_{opt} = 15 m^3/h$ and yields essentially a low difference to the measured head over the entire flow rate spectrum. The remaining difference between simulation and experiment is much less distinctive than the significant overestimation of the over load head by the hydraulically smooth surface simulation. However, a slight remaining under- and overestimation of the experiment EXP_R39/8 is discernible in part and over load, respectively. It is obvious that the inclusion of the roughness in the simulation model essentially compensates the overestimation of the head at over load that has been predicted by the hydraulically smooth model. However, a main source of head overestimation at over load remains, i.e. inaccurate representation of the separation due to the wall function. Therefore, the particular sand grain roughness height of $k_s = 140 \mu m$ is to be considered as an empirical correction factor of the simulation results. This conclusion is supported by the fact that $k_s = 140 \mu m$ is rather high compared to the measured values of $R_z \sim 39 \mu m$ and $R_a \sim 8 \mu m$.

that an inaccurate resolution of separation at the volute tongue leads to a head overprediction at over load and proceed with the wall function approach for cavitation flow.

### 2.3 Cavitating flow

Experiments and simulations for cavitating flow conditions are performed in order to analyse the effect of surface roughness on the $NPSH$. Figure 5 presents the evolution of the measured head depending on the $NPSH$ for four flow rates exemplarily for the case EXP_R13/3.

A sharp head drop is observed in part and nominal load, whereas the head decreases more smoothly in over load. The shape of the head drop curves from steady state simulations [10] is in qualitative agreement with the experiments. The $NPSH_{3\%}$ curve is depicted in figure 6. The standard deviation of the measurements is presented if the error bar exceeds the symbol size.

While the single-phase flow experiments reveal that the surface roughness has a considerable influence on the head (cf. figure 4), for the measured $NPSH_{3\%}$ the influence of the roughness on the $NPSH_{3\%}$ is minor (note also the scale difference of the ordinate in figure 4 and 6). An explanation of the minor impact of the volute roughness on the $NPSH_{3\%}$ curve is that the evaluation of the $NPSH_{3\%}$ value (equation 6) is based on the total pressure on the suction side only.

**Figure 5.** Measured head dependent on the net positive suction head for case EXP_R13/3.

**Figure 6.** $NPSH_{3\%}$ characteristics dependent on the surface roughness.
Numerical and experimental investigation of the cavitating flow in a low specific speed centrifugal pump and assessment of the influence of surface roughness on head prediction — 7/9

Therefore, the pressure losses on the pressure side and particularly in the volute due to roughness have a minor effect on the $NPSH_{3\%}$. Also the $NPSH_{3\%}$ curve evaluated from the simulation results does essentially not deviate between hydraulically smooth and rough walls. A slight underestimation of the measured $NPSH_{3\%}$ values in part and nominal load is observed. The steep increase towards over load is also distinctive in the simulation results, but occurs at a lower flow rate than in the experiment and results in an overestimation of the measured $NPSH_{3\%}$ values by the simulation at over load.

For a more detailed analysis of the cavitating flow the simulation results are analysed concerning the location of the cavitation zones in figure 7. The cavitation zones appear essentially stationary, and this fact is discussed further below. While the cavitation zones are mainly located at the suction side of the blades leading edge in part load, they move towards the pressure side with increasing flow rate (nominal load) due to a decreasing angle of incidence relative to the leading edge. At nominal load cavitation occurs on both, suction and pressure side. For part load conditions the impeller flow channels are uniformly filled, whereas the shape of the cavitation zones changes in circumferential direction from one flow channel to the other for nominal load depending on the position of the impeller relative to the volute tongue. At high flow rates ($1.3 \, Q_{opt}$) cavitation is observed at the tongue of the volute prior to cavitation at the blades, cf. figure 7c. As a consequence, according to the simulation results, the steep increase of $NPSH_{3\%}$ towards over load (cf. figure 6) is affected by the presence of vapour regions within the volute which is in agreement with the remarks of Gülich [13]. The steep rise of $NPSH_{3\%}$ in the simulation occurs at lower flow rates than in the measurement. However, the qualitative appearance of the $NPSH_{3\%}$ curve, in particular the presence of a steep rise towards over load, is obvious in both, experiment and simulation results. In the present study only sheet cavitation is observed in the simulation results at the blades as well as at the volute tongue, while also unsteady cloud cavitation is expected in the cavitation and Reynolds number regime in low specific speed centrifugal pumps. While in the case of cloud cavitation the unsteady nature of periodically detaching vapour clouds from a hydrofoil originates from the formation of a reentrant jet, the unsteady behaviour of cavitation at the volute tongue is also enforced from the changing flow angle due to the blade movement relative to the volute tongue [22]. Experiments of Bachert et al. [22] reveal the occurrence of periodic cloud cavitation at the volute tongue of a centrifugal pump ($n_q = 26 \, \text{min}^{-1}$), which is a very erosive type of cavitation as reported by Dular et al. [23]. However, periodic cloud cavitation is not captured by the present simulation model.

Summarising, the $NPSH_{3\%}$ deviation between experiment and simulation and particularly the rise of the $NPSH_{3\%}$ at too low flow rates in the simulation are supposed to originate from the following error sources:

- Although the neglect of axial thrust balance holes has essentially no effect for single-phase flow head evaluation, it may have a significant influence for the $NPSH_{3\%}$ evaluation: flow rate through the impeller increases due to leakage flow leading to a larger flow angle at the blade leading edge, whereas the volume flow through the volute does not change.

- Standard values for the cavitation model parameters, $F_{vap}$ and $F_{cond}$, have been utilised (cf. section 1.2). These parameters may have to be optimised for the particular flow situation in low specific speed pumps. This assumption is supported by the fact that cavitation zones appear essentially steady, while also unsteady cloud cavitation is expected.

- In addition to the parameter optimisation, a local reduction of the mixture eddy viscosity may lead to unsteady separation of the cavitation zones as shown by Coutier Delgosha et al. [24]. Although Zwart et al. [1] followed
Numerical and experimental investigation of the cavitating flow in a low specific speed centrifugal pump and assessment of the influence of surface roughness on head prediction — 8/9

3. CONCLUSIONS

3D CFD simulations as well as in-house experiments of a low specific speed centrifugal pump with varying surface roughness for single-phase and cavitating flow conditions are performed. Measured single-phase data show a trend of decreasing head with increasing surface roughness of the volute towards over load. For a reliable prediction of the head in the entire operation range of the pump the over load flow separation at the volute tongue should be captured by a low-Reynolds number wall approach and a correspondingly fine near-wall grid as well as by the simultaneous consideration of the surface roughness of the volute.

Concerning pump operation in cavitating flow the influence of surface roughness on $N P S H_{3\%}$ is minor in the experiments as well as in the simulation results. Although the measured $N P S H_{3\%}$ characteristics are slightly underestimated in part and nominal load and overestimated in over load, a distinctive and steep slope of the $N P S H_{3\%}$ curve is observed in both, the experiments and simulations for over load. Simulation results for this operation range confirm that the location of the cavitation zones switches from the impeller blade leading edges to the volute tongue.

The simultaneous application of low-Reynolds number wall treatment and roughness model, an extension of the grid study from single-phase flow to cavitating flow, the consideration of geometry details as axial thrust balance holes, the optimisation of the cavitation model parameters for the particular flow situation in low specific speed pumps as well as a modification of the mixture eddy viscosity following Coutier-Delgosha et al. [24] are suggested for future numerical research.

ACKNOWLEDGMENTS

The authors wish to gratefully acknowledge the financial support by the Verband Deutscher Maschinen- und Anlagenbau e.V. (VDMA), “Allianz Industrie Forschung” (AiF) as well as the Kompetenzzentrum Hydraulische Strömungsmaschinen (KHS) of the Ruhr-Universität Bochum.

NOMENCLATURE

Roman symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c_{eq}$</td>
<td>Equivalence coefficient</td>
<td>[-]</td>
</tr>
<tr>
<td>$D$</td>
<td>Diameter</td>
<td>[m]</td>
</tr>
<tr>
<td>$F$</td>
<td>Empirical parameter</td>
<td>[-]</td>
</tr>
</tbody>
</table>

Greek Characters

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha$</td>
<td>Volume fraction</td>
<td>[-]</td>
</tr>
<tr>
<td>$\Delta$</td>
<td>Delta operator</td>
<td>[-]</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Dynamic viscosity</td>
<td>[kg m$^{-1}$ s$^{-1}$]</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic viscosity</td>
<td>[m$^2$ s$^{-1}$]</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
<td>[kg m$^{-3}$]</td>
</tr>
</tbody>
</table>

Subscripts

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$3%$</td>
<td>Variable is related to a head drop of 3%</td>
<td>[-]</td>
</tr>
<tr>
<td>$B$</td>
<td>Bubble</td>
<td></td>
</tr>
<tr>
<td>$cond$</td>
<td>Condensation</td>
<td></td>
</tr>
<tr>
<td>$ES$</td>
<td>Evaluation surface</td>
<td></td>
</tr>
<tr>
<td>$1,2$</td>
<td>1, 2</td>
<td></td>
</tr>
<tr>
<td>$i,j$</td>
<td>Coordinate index</td>
<td></td>
</tr>
<tr>
<td>$k$</td>
<td>Phase</td>
<td></td>
</tr>
<tr>
<td>$l$</td>
<td>Liquid</td>
<td></td>
</tr>
<tr>
<td>$m$</td>
<td>Mixture</td>
<td></td>
</tr>
<tr>
<td>$nuc$</td>
<td>Nucleation site</td>
<td></td>
</tr>
<tr>
<td>$opt$</td>
<td>Best efficiency point</td>
<td></td>
</tr>
<tr>
<td>$t$</td>
<td>Turbulent</td>
<td></td>
</tr>
<tr>
<td>$v$</td>
<td>Vapour</td>
<td></td>
</tr>
<tr>
<td>$vap$</td>
<td>Vaporisation</td>
<td></td>
</tr>
</tbody>
</table>

Abbreviations and denominations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>3D</td>
<td>Three dimensional</td>
</tr>
<tr>
<td>BSL</td>
<td>Baseline</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational fluid dynamics</td>
</tr>
<tr>
<td>CP1.2</td>
<td>Centrifugal pump</td>
</tr>
<tr>
<td>CWF</td>
<td>coarse grid, wallfunction</td>
</tr>
<tr>
<td>ES</td>
<td>Evaluation surface</td>
</tr>
<tr>
<td>EXP_R39/8</td>
<td>Experiments with an untreated volute</td>
</tr>
<tr>
<td>EXP_R13/3</td>
<td>Experiments with a coated volute</td>
</tr>
<tr>
<td>FLRe</td>
<td>fine grid, low-Reynolds number approach in the volute</td>
</tr>
<tr>
<td>FWF</td>
<td>fine grid, wall function</td>
</tr>
<tr>
<td>NPSH</td>
<td>Net positive suction head</td>
</tr>
</tbody>
</table>
Numerical and experimental investigation of the cavitating flow in a low specific speed centrifugal pump and assessment of the influence of surface roughness on head prediction — 9/9

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NPSHR</td>
<td>Net positive suction head required</td>
</tr>
<tr>
<td>SIM_CWF_SMOOTH</td>
<td>Simulation results with grid CWF and hydraulically smooth walls</td>
</tr>
<tr>
<td>SIM_FLRe_SMOOTH</td>
<td>Simulation results with grid FLRe and hydraulically smooth walls</td>
</tr>
<tr>
<td>SIM_CWF_k_s,140</td>
<td>Simulation results with grid CWF and rough walls</td>
</tr>
<tr>
<td>SST</td>
<td>Shear stress transport</td>
</tr>
<tr>
<td>VoF</td>
<td>Volume of Fluid</td>
</tr>
</tbody>
</table>

REFERENCES


