INVESTIGATION ON THE INFLUENCE OF AN INDUCER ON THE TRANSPORT OF SINGLE AND TWO-PHASE AIR-WATER FLOWS BY CENTRIFUGAL PUMPS

Michael Mansour
Graduate Research Assistant
Laboratory of Fluid Dynamics & Technical Flows
University of Magdeburg “Otto von Guericke”
Magdeburg, 39106, Germany

Trupen Parikh
Master Student
Laboratory of Fluid Dynamics & Technical Flows
University of Magdeburg “Otto von Guericke”
Magdeburg, 39106, Germany

Sebastian Engel
Graduate Research Assistant
Laboratory of Fluid Dynamics & Technical Flows
University of Magdeburg “Otto von Guericke”
Magdeburg, 39106, Germany

Bernd Wunderlich
Scientific Assistant
Laboratory of Fluid Dynamics & Technical Flows
University of Magdeburg “Otto von Guericke”
Magdeburg, 39106, Germany

Dominique Thévenin
Faculty Dean and Chair Professor
Faculty of Process Engineering
Laboratory of Fluid Dynamics & Technical Flows
University of Magdeburg “Otto von Guericke”
Magdeburg, 39106, Germany

Michael Mansour received his B.S. and M.S. degrees in Mechanical Engineering from Helwan University in Cairo, Egypt. In 2009, he joined the Mechanical Engineering department at Helwan University as Research and Teaching Assistant. Afterwards, he started his work as a research assistant and began his Ph.D. degree in 2015 under the supervision of Prof. Dominique Thévenin at the Laboratory of Fluid Dynamics & Technical Flows at the Otto von Guericke University Magdeburg, Germany. Michael’s research mainly focuses on experimental and numerical investigations of gas-liquid two-phase flows in centrifugal pumps and diverging channels.

Trupen Parikh completed his B.E. in Mechanical engineering at NMAMIT, Nitte, India. In 2014, he went on to work as a Junior Mechanical engineer in India for a year. Later, he joined as a master student in Chemical and Energy engineering at the Otto von Guericke University Magdeburg. He is currently pursuing his Master study under the supervision of the Ph.D. candidate Michael Mansour and Prof. Dominique Thévenin at the Laboratory of Fluid Dynamics & Technical Flows at the Otto von Guericke University Magdeburg, Germany. His thesis mainly focuses on numerical analysis of two-phase gas/liquid in turbomachinery.

Sebastian Engel received his B.Sc. and M.Sc. degrees in Mechanical Engineering from the Otto von Guericke University Magdeburg, Germany. Since 2015, he is working as a research assistant, pursuing his Ph.D. degree under the supervision of Prof. Dominique Thévenin, Head of the Laboratory of Fluid Dynamics and Technical Flows, Otto von Guericke University Magdeburg, Germany. Sebastian also is a student of the graduate school IMPRS of the Max-Planck-Institute for Dynamics of Complex Technical Systems. His research focuses on medical engineering of centrifugal pumps and other blood-wetted devices. He is a CFD expert and employs a multitude of methods, such as CFD-based design optimizations, model benchmarking, and non-Newtonian flows. His contribution is supported by the center of dynamic systems (CDS), which is funded by the EU-program ERDF.
Bernd Wunderlich is doing research on the field of gas-liquid two-phase flows since his diploma thesis in 1973 at the former Otto von Guericke University Magdeburg. He published a number of experimental studies concerning two-phase flows in radial centrifugal pumps. He received his Ph.D. degree from the same university in 1981. His research team had been given the research award of the Otto von Guericke University Magdeburg in 1981 for the excellent activities in this field. Since his retirement, he has been engaged with teaching by giving lectures and tutorials in Fluid Mechanics, Turbomachinery and Engineering Measurements. Further, he developed several measuring methods for special applications and registered more than 30 patents.

Dominique Thévenin is currently the Dean of the Faculty of Process Engineering at the Otto von Guericke University Magdeburg. Since 2002, he holds the Chair of the Laboratory of Fluid Dynamics and Technical Flows, including teaching and research. Prof. Thévenin started his scientific career in 1989 after completing his studies at École Polytechnique, France. Subsequently, in 1992, he received his Ph.D. degree in energy technology at the Ecole Centrale Paris, France. His research interests involve in particular Direct Numerical Simulation (DNS) applied to multiphase flows, reactive flows, sprays, laminar and turbulent flames. But he also considers medical flows, CFD-based optimization of turbomachines, as well as mixing and separation processes.

ABSTRACT

A joint numerical/experimental study has been carried out to understand the influence of an upstream inducer on the transport of single and two-phase air-water flows by a semi-open pump impeller of radial type. The inducer was found to have generally a slight influence on single-phase flow. For two-phase flows, the experiments show that the performance of the pump without inducer is strongly reduced when the gas volume fraction $\varepsilon$ exceeds 4%, due to the onset of large gas accumulations in the impeller. When the inducer is installed, the abrupt performance drop of the pump could be positively delayed to $\varepsilon = 7\%$ at part-load flow conditions. However, in overload flow conditions, the inducer shows only a slight enhancement, up to $\varepsilon = 5\%$, before a rapid deterioration occurs. Using numerical simulations, the flow details within the inducer have been investigated, considering a simplified 3D model. The presentation will discuss the impact of the employed numerical settings. Steady-state as well as transient simulations with different turbulence models (Realizable $k-\varepsilon$, $k-\omega$ SST, and Reynolds-Stress Model – RSM) have been compared. The steady-state simulations were done using a frozen-rotor approach with a Moving Reference Frame (MRF), while the transient approach was represented using a moving-mesh approach. Compared to other turbulence models, RSM was found to be able to capture more details of the flow, leading to an excellent agreement with experimental data, while the simpler turbulence model exhibit larger deviations. The numerical analysis can clarify the sudden fall in inducer's efficiency at overload conditions; the flow separates and forms strong, axially propagating vortices within the inducer, resulting in negative pressure change across the inducer.

INTRODUCTION

Centrifugal pumps are widely employed in numerous technical and industrial applications due to their simple design and broad flexibility for demand requirements. They can excellently transport single-phase flows of liquids with highly reliable performance. For two-phase flows, only expensive and specially-designed multiphase pumps can handle high gas contents (50% – 100%), such as multiple-stage Electrical Submersible Pumps (ESP) [1,2] or twin-screw pumps [3,4]. However, when transporting two-phase gas-liquid mixtures by a single-stage centrifugal pump, the pumping performance deteriorates abruptly, even at very low gas volume fractions (about 1%). This occurs due to the high tendency of the gas for rapid accumulation along the blades of the impeller, preventing the energy transfer from the blades to the fluid being transported. Subsequently, multiple undesirable pumping problems can arise in the pump.

Firstly, at gas flowrates between 4% and 6%, impeller channels can be completely blocked by the gas in most single-stage radial centrifugal pumps, which is well-known as the “gas-locking” phenomenon [5–10]. Secondly, the flow can be totally stopped, preventing any discharge flow by the pump at about 7% to 10% gas contents, which is known as “pump break-down” [7–11]. Furthermore, severe instabilities and intense system vibrations can develop under specific flow conditions as a result of the continuous formation and discharging of large gas accumulations. This condition is usually called “pressure surging” or “pump surging”, under which all pump parameters (flowrate, head, efficiency) oscillate irregularly between two different operational points [9,10,12–16]. These problems can readily occur in many engineering applications and systems that involve pumping of two-phase flows, including for example, petroleum and natural gas production [5,17], geothermal power plants [18], nuclear power plants [8], chemical processes, paper industry, or wastewater treatment [11].

2

Copyright© 2019 by Turbomachinery Laboratory, Texas A&M Engineering Experiment Station
Visualization studies showed that the pump break-down occurs as soon as the accumulated gas grows to reach the outer diameter of the impeller [19]. The two-phase flow regimes in the impeller channels were identified and classified to distinct patterns in the literature, including mainly bubble flow, agglomerated bubble flow, unstable (alternating) pocket flow, and segregated (separated) flow [9,10,12,19–21]. The details and the corresponding characteristics of each flow regime can be found in the recent study of [9]. The different two-phase regimes are also shown in this paper and briefly discussed.

The presence of large separation zones in the flow was found to be the main reason for the development of huge gas accumulations in two-phase flows [22]. The accumulated gas can be reduced when the turbulence levels are increased near the accumulations [22]. Accordingly, the two-phase pumping capacity can be always improved by increasing the rotational speed of the pump, leading to higher turbulence and more dispersed gas bubbles (i.e., more homogeneous mixtures). This allows the impeller to transport higher gas contents [16,20,22,23].

Due to the leakage flow that occurs within the gap of semi-open impellers, they show usually an improved resistance to gas accumulation [10,11,24–26]. The secondary flow moving across the blades in the tip-clearance gap disturbs the gathered gas pockets, delaying the degradation of pump performance [10]. However, semi-open impellers lead to undesirably strong performance hysteresis, when different experimental procedures are used to set the desired flow conditions. This means that the pump performance depends on the history of setting the gas and liquid flowrates. Adding the gas flow starting from the no-gas condition or decreasing it from an initially high value, the pump performance can be different for identical operating conditions. The reason is the former accumulation of gas pockets, which can persist on the impeller blades, even if the gas flow is decreased to the rate, where smaller or no accumulation is expected [9,10]. Additionally, some studies showed that the two-phase pumping performance can be improved by slightly increasing the tip-clearance gap of the impeller [9,11,27], although this causes a simultaneous drop in the single-phase performance of the impeller. With a standard gap, semi-open impellers can handle gas volume fractions up to approximately 4%, before the subsequent sudden performance drop [9,11]. With twice the gap width, the accompanied higher turbulence and the increased secondary flow produce higher resistance of gas accumulation, which is always preferred when considering high gas contents (4% to 7%, [9]). The increased gap was found also very effective to damp the flow instabilities and to eliminate the performance hysteresis [9].

Inducers are widely used in the industry along with centrifugal pumps due to their ability to enhance the suction conditions, keeping the inlet pressure well above the vapor pressure in order to avoid cavitation. They are simply axial flow impellers that can be mounted upstream of the main impeller. The number of inducer blades is usually low (typically 2-4 helical blades) to ensure low inlet solidity [28]. They are mostly installed for high-flowrate pumps to reduce the required net positive suction head (Sulzer [28,29]). Only insignificant changes occur in the single-phase head of centrifugal pumps when an inducer is used with the main impeller [9]. Nevertheless, inducers were found to have several positive effects on the transport of two-phase flows by centrifugal pumps [9,11]. For example, it was shown by Mansour et al. [9] that installing the inducer can improve the mixing of the phases before the impeller inlet, which moderately damps the flow instabilities and significantly decreases the performance hysteresis between different procedures of setting the desired conditions. Further, the improved phase mixing provided by the inducer results generally in improved two-phase pumping capability. However, the positive influence of the inducer is much more pronounced in part-load flow conditions, compared to a very limited improvement in overload flow conditions. Here, part-load and overload conditions refer to flowrates being lower or higher than the rated (nominal) flowrate of the pump, respectively. The sudden performance drop of the pump can be positively delayed to $\varepsilon = 7\%$ at part-load flow conditions. However, in overload flow conditions, the inducer can delay the steep performance drop only from $\varepsilon = 4\%$ to $\varepsilon = 5\%$. The interior flow details and the physical mechanisms explaining the dissimilar effects of the inducer on two-phase pumping performance in part-load and overload conditions are still not completely clear.

Filling this gap is the main objective of the present work by studying the flow details of the inducer numerically, after summarizing the main observations of Mansour et al. (2018b) concerning the influence of installing the inducer on the single and two-phase pump performances. Additionally, the numerical results are compared with a measured performance for the inducer. The same inducer geometry used in the work of Mansour et al. (2018b) is employed as well in this present work. In accordance with our previous studies [9,10], a low rotational speed of 650 rpm is kept, which is necessary to avoid any damage of the optically transparent impeller made of acrylic glass in the experiments. A simplified 3D simulation domain is built, focusing specifically on the inducer flow. Covering from part-load to overload conditions, the full range of the pump flowrate is considered. Steady simulations with different turbulence models (Realizable $k-\varepsilon$, $k-\omega$ SST, and Reynolds-Stress Model – RSM) are compared using the Moving Reference Frame (MRF) approach, together with unsteady simulations with the moving-mesh approach. The present analysis is useful to understand the operating range of inducers when used for improving the performance of two-phase pumping. Additionally, the comparisons between numerical and experimental results are useful for selecting suitable numerical models for such a complex flow.

Copyright© 2019 by Turbomachinery Laboratory, Texas A&M Engineering Experiment Station
EXPERIMENTAL TEST-RIG

Figure 1 shows the entire closed loop of the experimental set-up. A large tank employed with an air-release pipe, allow the gas phase to exit the tank. The water flowrate is measured separately by an electro-magnetic flow-meter (Endress+Hauser Promag 30F, with ± 0.5 % RD accuracy). The water flowrate is controlled by a motorized gate valve installed on the discharge line. The inlet air-line is equipped with a mass flow-meter (Bronkhorst F-113AC-HD-55-V, with ± 0.5 % RD plus ± 0.1 % FS accuracy) and a throttle valve to control the flow. The air is injected in the center of the suction pipe near the pump inlet through a gas distribution nozzle. The pressure is measured across the inducer explicitly by absolute pressure sensors (Sensotec Z, with ± 0.25 % FS accuracy). The inlet pressure is used to calculate the air volume flowrate and the air volume fraction. The flow temperature is measured near the injected air by a temperature sensor (Pt100 Industrial Sensor Probe, Class B, with ± 0.3 K maximum absolute error). A variable-speed electric motor (5.7 kW, with 3000 maximum rpm) is employed to drive the pump. A frequency controller and an analog tachometer are used to set and measure the rotational speed, respectively. As shown, the experimental set-up is not sealed, where the single-phase liquid flow is saturated with air (not deaerated). However, the pump head generated by the impeller is relatively small at 650 rpm. Therefore, removing the dissolved air would not make much difference in the pump performance.

Figure 1. Schematic sketch of the experimental test-rig.

The geometrical details of the pump, the impeller, and the inducer are shown in Figure 2 and Table 1. As can be seen, the employed inducer has 3 helical blades and is installed directly ahead of the main semi-open impeller. The inducer has a high efficiency within the range of $Q/Q_{\text{opt}} = 0.6-0.8$, where $Q_{\text{opt}}$ is the best efficiency flow of the impeller. The “radial” tip-clearance gaps of the impeller and the inducer are $S_1/b_2 = 2.5\%$ and $S_i/b_2 = 12.5\%$, where $b_1$ and $b_2$ are the impeller blade inlet and outlet widths as shown in Figure 2. Since most pump components are made of acrylic glass, a low rotational speed of 650 rpm was kept for all experiments, preventing cavitation and strong vibrations occurring at higher speeds. As a consequence, the specific speed of the pump is only approximately $N_q = 21 \left( N_x = 1084 \right) \text{ min}^{-1}$, as defined by Equation (1):

$$N_q = \frac{n \sqrt{Q}}{H^{3/4}}$$

where $n$ is the rotational speed of the pump in (rpm), $Q$ is the volume flowrate in (m$^3$/s) and $H$ is the pump head in (m). The experiments were done for a suction pressure within the range of 1.14-1.28 bar. In our previous study [9], the performance of the impeller with and without inducer was analyzed and compared together with the effect of the tip-clearance gap of the semi-open impeller.
Figure 2. Geometrical details of the pump, the impeller, and the inducer.
Table 1: Geometrical dimensions of the impeller and the inducer

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Symbol</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impeller inlet blade angle</td>
<td>$\beta_1$</td>
<td>24°</td>
</tr>
<tr>
<td>Impeller outlet blade angle</td>
<td>$\beta_2$</td>
<td>24°</td>
</tr>
<tr>
<td>Impeller blade number</td>
<td>$n_{b,imp}$</td>
<td>6</td>
</tr>
<tr>
<td>Impeller tip clearance gap</td>
<td>$S/b_2$</td>
<td>2.5%</td>
</tr>
<tr>
<td>Inducer blade number</td>
<td>$n_{b,ind}$</td>
<td>3</td>
</tr>
<tr>
<td>Hub diameter to tip ratio</td>
<td>$D_h/D_i$</td>
<td>41.85%</td>
</tr>
<tr>
<td>Inducer tip clearance gap</td>
<td>$S_i/b_2$</td>
<td>12.5%</td>
</tr>
<tr>
<td>Hub solidity</td>
<td>$\sigma_h$</td>
<td>2.836</td>
</tr>
<tr>
<td>Tip solidity</td>
<td>$\sigma_t$</td>
<td>1.807</td>
</tr>
<tr>
<td>Inducer area solidity</td>
<td>$\sigma_a$</td>
<td>23.326%</td>
</tr>
<tr>
<td>Sweep angle</td>
<td>$\varphi$</td>
<td>29.4°</td>
</tr>
<tr>
<td>Blade axial length</td>
<td>$L_b/d$</td>
<td>1.0</td>
</tr>
</tbody>
</table>

SIMULATION DOMAIN

The numerical investigations focus mainly on the inducer performance and the corresponding flow details. Accordingly, a simplified 3D model of the inducer was derived and employed in the CFD analysis (CFD for Computational Fluid Dynamics) to reduce the cost of the numerical simulations, excluding the downstream domain containing the impeller. The details and the dimensions of the considered simulation domain are shown in Figure 3. All main dimensions are given as a function of the suction pipe diameter ($d$). Based on the location of the experimental pressure sensors, two pressure probes are used similarly in the simulation across the inducer. The pressure sensor 2 is set at the end of the inducer blades because it was not possible experimentally to move it further downstream the inducer since the impeller is directly installed behind the inducer (see Figure 2). The domain is split into two equal parts, i.e. a stationary domain (before the inducer) and a rotating domain (around the inducer). The pipe wall’s tangential speed is kept zero in the whole domain in respect to the stationary reference frame, allowing only the inducer body to rotate relative to the flow.

![Figure 3. Details and dimensions of the 3D simulation model.](image-url)
NUMERICAL MODELING

Model description

The commercial software Siemens STAR-CCM+ V13.02 was used in the present study to perform the CFD simulations. A Moving Reference Frame (MRF) approach is applied for all single-phase simulations. The MRF approach is very effective for modeling the time-averaged properties of a moving flow field, by moving the observer coordinates. It can basically model the motion by steady-state simulations, yet it eliminates the need for rotating the inducer body or moving the mesh, which limits the cost of computation. In MRF simulations, a constant grid flux (i.e. a Coriolis force that is induced by the rotation) is imposed in the source term of appropriate conservation equations for the defined rotating domain where the inducer is placed.

For transient simulations, the moving-mesh approach was applied, considering now the rotation of the inducer domain. A timestep of 0.256 ms was set, corresponding to a one-degree angle rotation. Additionally, 40 inner iterations were used to confirm the convergence of every iteration by at least a drop of three magnitudes of all residuals. The simulations were stopped after simulating 0.5 seconds of physical time, corresponding to more than 5 complete revolutions of the inducer, which are found sufficient to reach statistically steady conditions.

Numerical mesh

A polyhedral mesh was generated for the whole simulation domain, considering an in-place interface between the stationary and the rotating domain. A total of 8 prism layers were generated at all walls, for an accurate resolution of the boundary layer flow. To confirm that the numerical results have low dependence on the mesh, a mesh-convergence study was first performed, considering different mesh resolutions. The mesh study was carried out at $Q_w/Q_{opt} = 1$ and at $Q_w/Q_{opt} = 0.5$ using MRF and Reynolds Stress Model (RSM) for turbulence modeling. The details of the considered grids for the mesh-convergence study are listed in Table 2. The height of the first cell near the wall was set low enough, leading mostly to an average non-dimensional wall distance ($y^+$) lower than 1 to avoid the need for wall models. The numerical results for the pressure change across the inducer are shown in Figure 4, where no significant changes can be seen in the pressure values when the mesh is refined up to 5.1 million cells (mesh 5). Therefore, mesh 4 with 3.25 million cells, was assumed to have a mesh-independent solution, and thus kept for all further simulations. The numerical results of the selected mesh show also excellent agreement with the experimental data for the whole flowrate range as disused later. A sample view of mesh 4 is given in Figure 5.

Table 2. Details of different grids used for the mesh-independence study.

<table>
<thead>
<tr>
<th>Mesh #</th>
<th>No. of cells (millions)</th>
<th>Average wall $y^+$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.14</td>
<td>1.95</td>
</tr>
<tr>
<td>2</td>
<td>0.46</td>
<td>0.94</td>
</tr>
<tr>
<td>3</td>
<td>1.4</td>
<td>0.48</td>
</tr>
<tr>
<td>4</td>
<td>3.25</td>
<td>0.31</td>
</tr>
<tr>
<td>5</td>
<td>5.1</td>
<td>0.25</td>
</tr>
</tbody>
</table>

Figure 4. Comparison of the numerical results using different meshes at $Q_w/Q_{opt} = 1$.  

Copyright© 2019 by Turbomachinery Laboratory, Texas A&M Engineering Experiment Station
RESULTS AND DISCUSSION

Experimental single-phase pump performance curves

Some important observations and findings from our previous study [9] regarding the overall influence of the inducer on the pump performance, are summarized in this section. Figure 6 presents the normalized performance curves of the pump with and without inducer when transporting a single-phase flow, where $\Upsilon$ is the pump specific delivery work (see Equation (2)), $Q_w$ is the volume flowrate, $\Upsilon_{opt}$ is the optimal specific delivery work and $Q_{w, opt}$ is the optimal volume flowrate of the pump. Here, the optimal conditions of the pump without inducer corresponding to the maximum efficiency point are used for normalization in Figure 6. As shown, the influence of the inducer is generally insignificant for single-phase flow. However, the pump performance with the inducer is slightly increased in part-load conditions and slightly decreased in overload conditions. The reasons for these positive and negative changes in the performance curve, when the inducer is installed, are analyzed later by the CFD simulations.

Experimental two-phase pump performance curves

For two-phase experiments the pump performance is calculated in terms of the specific delivery work ($\Upsilon$) which is defined by Equation (2), considering also the compression work of air, assuming isothermal process across the pump.

$$\Upsilon = \frac{1 - \tilde{\mu}}{\rho_w} (P_D - P_s) + \tilde{\mu} R T \ln \left( \frac{P_D}{P_s} \right) + \frac{1}{2} (V_D^2 - V_s^2) + g(z_D - z_s)$$  \hspace{1cm} (2)

where $\tilde{\mu}$ is the air mass fraction (also called the mixture quality), $\rho_w$ is the water density, $P_D$ is the discharge pressure, $P_s$ is the suction pressure, $R$ is the gas constant of air, $T$ is the flow temperature, $V_D$ is the discharge superficial flow velocity, $V_s$ is the suction superficial flow velocity, $g$ is the gravitational acceleration, $z_D$ is the discharge elevation and $z_s$ is the suction elevation. The air mass fraction ($\tilde{\mu}$) is defined by Equation (3), where $m_a$ and $m_w$ are the mass flowrates of air and water, respectively and $\rho_a$ is the air density. The superficial velocities in the suction and the delivery pipes ($V_s$ and $V_D$) are calculated from Equation (4) and (5), respectively, based on the continuity balance, where $A_s$ and $A_D$ are the suction and the discharge areas, respectively. The complete derivation of Equation (2) can be found in our previous study [9].
The performance of the pump was obtained for different gas volume fractions $\varepsilon$, which is defined by Equation (6), representing the fraction of the air volume flowrate to the total volume flowrate ($Q_t$). Here, $Q_t$ is the sum of $Q_a$ and $Q_w$, which are the air and water volume flowrates, respectively.

$$\varepsilon = \frac{Q_a}{Q_t} = \frac{Q_a}{Q_a + Q_w}$$

Figure 7 and Figure 8 show the two-phase performance curves of the pump without and with inducer, respectively, in terms of the normalized specific delivery work ($Y/Y_{opt}$). As shown in Figure 7, the operating range of the pump becomes progressively narrower by increasing the gas volume fraction ($\varepsilon$). In part-load conditions, pump break-down occurs, where the pump runs but completely loses the ability to transport the mixture. Further, starting from $\varepsilon = 4 \%$, the slope of the performance degradation is much steeper compared to the range of $\varepsilon = 1 \sim 3 \%$. In overload conditions, considerable instabilities in the pump delivery and strong system vibrations occur, as indicated by “Surging” on Figure 7 and Figure 8. The data points indicating surging are shown by gray-filled symbols in Figure 7 and Figure 8. As shown in Figure 8, by installing the upstream inducer, the pump performance is noticeably improved in the part-load range, where the pump keeps a very good performance up to a gas volume fraction of $\varepsilon = 7 \%$, delaying the abrupt performance deterioration. Moreover, the inducer can also reduce the conditions corresponding to pump surging and pump break-down. However, the performance is only improved slightly in overload conditions up to $\varepsilon = 4 \sim 5 \%$. The reasons for the disparity in the effect of the inducer along the performance curves are explained and discussed later by simulating the flow across inducer.
As stated previously, the whole pump body and the impeller were manufactured from acrylic glass to allow for high-quality flow visualization. A high-speed camera (Model: Imager pro HS 4M CCD) with a resolution of 2016 x 2016 pixels was employed to acquire the flow images. Four LED lamps were installed around the pump casing to illuminate the flow, leading to bright gas-liquid interfaces and making the bubbles and the accumulated gas clearly visible. In our studies, the observed two-phase flow regimes are classified based on previous literature to:

- **Bubbly flow regime**: As can be seen from Figure 9a, the gas bubbles are well dispersed everywhere in the impeller passages with nearly homogeneous density. In this regime, no significant interaction occurs between the bubbles.

- **Agglomerated bubbles regime**: The bubbles start to interact with each other by coalescence, forming larger bubbles near both sides of the blades, usually with more bubbles are observed near the suction side, as shown in Figure 9b.

- **Alternating pocket flow regime**: In this regime, a large gas pocket appears near to the entrance part of some blades (see Figure 9c), mostly on the suction side, with large size oscillations and unstable appearance and disappearance. The pump performance is very unstable.

- **Pocket flow regime**: A large (stable) air pocket stands on the suction side of all blades as shown in Figure 9d, with only slight size changes. In several cases, a remarkable drop of the corresponding pump performance is observed once the pocket appears.

- **Segregated flow regime**: The gas pocket is very huge in this regime and is long enough cover the whole length of the blades. This regime could not be observed in the semi-open impeller, while it could be observed only in the closed impeller studied in one of our previous studies [12].

![Figure 8. Measurements of the two-phase performance curves of the pump with the inducer.](image)
Comparing now the flow regimes for the impeller with and without the inducer, it can be generally identified that the use of the inducer results in more bubble dispersion in the impeller, in addition to the improved phase mixing at the impeller inlet. The phase mixing eventually delays the transition of the agglomerated bubble regime to the gas pocket regimes by avoiding coalescence of bubbles thus delaying the pump performance degradation at underload and optimal conditions. Nevertheless, the flow regimes are overall comparable for the two cases with similar flow features.
Inducer performance

Firstly, a comparison of the numerical results considering different turbulence models is shown together with the measured single-phase performance of the inducer. Figure 10 compares the numerical results of different turbulence models with the experimental data, considering the realizable $k - \varepsilon$ [30], $k - \omega$ SST [31], and Reynolds-Stress Model – RSM with quadratic pressure-strain term [32,33]. The complete formulation of all used models can be found in the user manual of STAR-CCM+ [34]. Here, the realizable $k - \varepsilon$ was considered rather than the standard $k - \varepsilon$, since it can more accurately predict separated and recirculating flows [35], sometimes also better than the $k - \omega$ SST model [36]. The single-phase performance comparison shown in is presented in terms of head coefficient ($\psi$) which represents the pressure change across the inducer ($P_2 - P_1$) and defined by Equation (7). Here, $Q_{\text{opt}}$ stands for the optimal flowrate of the pump with inducer, $u_t$ is the tip seed of the inducer blades.

\[
\psi = \frac{p_2 - p_1}{\rho w u_t^2}
\]  

(7)

Compared to the experiments, it is clear from Figure 10 that most models can generally predict the inducer performance, particularly near the optimal flowrate of the pump. However, at $Q_{w}/Q_{\text{opt}} = 0.4$, the realizable $k - \varepsilon$ and the $k - \omega$ SST models deviate obviously from the measured data. Additionally, in overload conditions, all first-order turbulence models (i.e. the realizable $k - \varepsilon$ and the $k - \omega$ SST models) slightly overestimate the experimental curve. On the other hand, RSM can predict the head coefficient very precisely, since it matches quite well the measured data along the whole flowrate range. Further, it can be seen that both the steady MRF and the transient moving mesh simulations lead to very similar values.

It can be seen in Figure 10 that the inducer performance is primarily negative for overload conditions. This explains the slight reduction of the single-phase pump performance when the inducer is installed, as shown previously in Figure 6. Additionally, this is identified as the main reason for the sudden reduction of the two-phase pumping performance when the pump is running with the inducer, as discussed later. Despite the generally negative pressure change across the inducer in overload conditions, it can still improve the two-phase pumping performance slightly up to $\varepsilon = 4 - 5 \%$, since the inducer can improve the mixing of the two phases. Now, to understand this dissimilar inducer performance along the flowrate rage, the flow is further analyzed by examining the velocity fields at radial and axial sections across the blades of the inducer. Figure 11 illustrates the location of the sections used for the analysis of the velocity fields. The axial section was set vertically in the middle of the pipe, while the radial section was placed at a distance of $x = 3.5d$ from the inlet surface of the simulation domain (approximately mid length of the blades).
Figure 12 presents the velocity fields in the axial section for the different turbulence models applied. The first, the second and the third columns correspond to part-load \((Q_w/Q_{opt} = 0.4)\), optimal \((Q_w/Q_{opt} = 1.0)\) and overload \((Q_w/Q_{opt} = 1.6)\) conditions, respectively. Again, both unsteady and steady simulations lead to very comparable results. Further, the flow streamlines are quite smooth in the longitudinal direction for optimal and overload conditions. Additionally, no significant differences can be seen in the velocity fields obtained by different turbulence models for optimal and overload conditions. However, tangential vortices occur for part-load conditions (at \(Q_w/Q_{opt} = 0.4\)), mostly upstream the inducer around its hub. These vortices can be in principle predicted by all turbulence models, but RSM can capture the vortices more accurately, with stronger streamline curvatures and more apparent structures compared to other models. Accordingly, the pressure change of RSM is very close to the measured value at \(Q_w/Q_{opt} = 0.4\), while the other models deviate.

Similarly, Figure 13 compares the velocity fields along the radial cut for different turbulence models and flowrates. In the radial section, only slight changes can be seen when either different turbulence models, or steady and unsteady simulations are compared. For optimal and part-load conditions, the flow is relatively smooth with some curved streamlines at part-load conditions due to the presence of tangential vortices in this case. For overload conditions, the simulation results reveal that the flow obviously separates and forms strong vortices near the blades. These vortices can be captured by all turbulence models, as shown in the last column of Figure 13.

Now, to investigate the inducer performance further, the Q-criterion is presented in Figure 14 similarly for part-load, optimal, and overload flow conditions using RSM. The Q-criterion is a three-dimensional, scalar indicator that can be used to visualize a vortex as a spatial region [37]. It can be used to obtain 3D views of the wake regions and the vortex structures in a specific flow. As can be seen in Figure 12, tangential vortices occur for part-load conditions mainly upstream of the inducer as shown also in Figure 12; they have generally a low influence on the inducer performance since the pressure change across the inducer is positive in this case.

For optimal flow conditions, weak flow separations occur at the leading edge of the blades and rapidly decay downstream, leading again to a smooth flow within the inducer. Additionally, some wake vortices are generated behind the trailing edge of the inducer, reducing slightly the inducer performance. However, Figure 14 reveals additionally that the flow separations occurring in overload conditions are propagating in the axial direction within the inducer along the whole length of the blades, resulting finally in very low performance. The presence of flow separation and strong vortices justify the negative pressure change across the inducer for overload conditions. Moreover, this negative pressure change will also result in an expansion of the gas phase at the impeller inlet, allowing it to occupy more space in the impeller channels, leading in the end to the abrupt drop of the overload performance occurring when the pump is running with the inducer. The present results confirm that the effective range of the inducer for enhancing two-phase pumping is limited mostly to part-load and near-optimal flow conditions.
Part-load conditions
\( \frac{Q_w}{Q_{opt}} = 0.4 \)

Optimal conditions
\( \frac{Q_w}{Q_{opt}} = 1.0 \)

Overload conditions
\( \frac{Q_w}{Q_{opt}} = 1.6 \)

Figure 12. Velocity fields along the axial section for different turbulence models and flowrates.
Part-load conditions
$Q_w/Q_{opt} = 0.4$

Optimal conditions
$Q_w/Q_{opt} = 1.0$

Overload conditions
$Q_w/Q_{opt} = 1.6$

Figure 13. Velocity fields along the radial section ($x = 3.5d$; mid blade length) for different turbulence models and flowrates.
Part-load conditions
$Q_{aw}/Q_{opt} = 0.4$

Optimal conditions
$Q_{aw}/Q_{opt} = 1.0$

Overload conditions
$Q_{aw}/Q_{opt} = 1.6$

Figure 14. 3D views of the vortex structures represented by Q-criterion (threshold value is $Q = 20000 \text{ s}^{-2}$) using steady-state RSM simulations for different flowrates.
CONCLUSIONS

In the present work, investigations have been done to study the influence of an upstream inducer on the transport of single and two-phase air-water flows by a centrifugal pump. The main conclusions can be summarized as follows:

- For single-phase flow, the inducer has only a slight influence on the pump performance curve, while for two-phase flows, the inducer is able in part-load flow conditions to delay the abrupt performance drop of the pump from \( \varepsilon = 4\% \) to \( \varepsilon = 7\% \). However, in overload flow conditions, the inducer shows only a slight enhancement, up to \( \varepsilon = 4 - 5\% \).

- A novel comparison for different turbulence models was done for the inducer flow using the steady-state MRF approach, revealing that RSM is able to match the measurements very accurately along the whole range of the pump flowrate. Other, first-order turbulence models (realizable \( k - \varepsilon \) and \( k - \omega \) SST) deviate moderately from the experimental points. This suggests indirectly that the RSM model can predict flow vortices and streamline curvatures more accurately compared to the other turbulence models.

- Comparing steady-state simulations using MRF and transient simulations using moving-mesh approach, no significant change could be found, showing that the steady MRF simulations are able to predict the inducer performance quite accurately for this configuration.

- Considering the inducer performance, a positive pressure change is observed in part-load conditions, while the pressure change is negative at overload. By simulating the flow across the inducer, the negative inducer performance occurring in overload conditions could be explained. A strong flow separation is visible in such conditions, forming large axially propagating vortices along the inducer blades.

- In addition to the strong flow separation, the negative pressure change occurring across the inducer in overload conditions will simultaneously allow the air to expand and occupy more space in the impeller channels, explaining the sudden drop of the overload performance occurring when the pump is running with the inducer.

The present study explains the different influences of the inducer on the two-phase pumping performance in part-load and overload conditions. It can only be used to improve two-phase pumping at part-load.

NOMENCLATURE

\[ A_D = \text{Discharge cross-sectional area} \quad (L^2) \]
\[ A_S = \text{Suction cross-sectional area} \quad (L^2) \]
\[ b_1 = \text{Blade inlet width} \quad (L) \]
\[ b_2 = \text{Blade inlet width} \quad (L) \]
\[ d = \text{Suction pipe diameter} \quad (L) \]
\[ g = \text{Gravitational acceleration} \quad (L \ T^{-2}) \]
\[ L = \text{Length of the simulation domain} \quad (L) \]
\[ \dot{m}_a = \text{Mass flowrate of air} \quad (M \ T^{-1}) \]
\[ \dot{m}_w = \text{Mass flowrate of water} \quad (M \ T^{-1}) \]
\[ n = \text{Rotational speed} \quad (T^{-1}) \]
\[ N_S = \text{Specific speed} \quad (m^3/s, m, rpm) \quad (T^{-1}) \]
\[ N_o = \text{Specific speed} \quad (USGPM, ft, rpm) \quad (T^{-1}) \]
\[ P_1 = \text{Upstream pressure} \quad (M \ L^{-1} \ T^{-2}) \]
\[ P_2 = \text{Downstream pressure} \quad (M \ L^{-1} \ T^{-2}) \]
\[ P_2 - P_1 = \text{Pressure change} \quad (M \ L^{-1} \ T^{-2}) \]
\[ P_D = \text{Pump discharge pressure} \quad (M \ L^{-1} \ T^{-2}) \]
\[ P_s = \text{Pump suction pressure} \quad (M \ L^{-1} \ T^{-2}) \]
\[ Q = \text{Volume flowrate} \quad (L^3 \ T^{-1}) \]
\[ Q_a = \text{Air volume flowrate} \quad (L^3 \ T^{-1}) \]
\[ Q_{\text{max}} = \text{Maximum volume flowrate} \quad (L^3 \ T^{-1}) \]
\[ Q_{\text{opt}} = \text{Optimal volume flowrate} \quad (L^3 \ T^{-1}) \]
\[ Q_t = \text{Total volume flowrate} \quad (L^3 \ T^{-1}) \]
\[ Q_w = \text{Water volume flowrate} \quad (L^3 \ T^{-1}) \]
\[ R = \text{Gas constant of air} \quad (L^2 \ T^{-2} \ O^{-1}) \]
\[ S = \text{Impeller tip-clearance gap} \quad (L) \]
\[ S_i = \text{Inducer tip-clearance gap} \quad (L) \]
\[ T = \text{Flow temperature} \quad (\Theta) \]
\[ V_D = \text{Discharge velocity} \quad (LT^{-1}) \]
\[ V_S = \text{Suction velocity} \quad (LT^{-1}) \]
\[ z_D = \text{Discharge elevation} \quad (L) \]
\[ z_S = \text{Suction elevation} \quad (L) \]
\[ \varepsilon = \text{Gas volume fraction} \quad (-) \]
\[ \mu = \text{Gas mass fraction} \quad (-) \]
\[ \rho_a = \text{Density of air} \quad (ML^3) \]
\[ \rho_w = \text{Density of water} \quad (ML^3) \]
\[ \Phi = \text{Specific delivery work} \quad (L^2T^{-2}) \]
\[ \Phi_{opt} = \text{Optimal specific delivery work} \quad (L^2T^{-2}) \]
\[ \psi = \text{Head coefficient} \quad (-) \]

CFD = Computational fluid dynamics
FS = Accuracy in percentage of full scale
MRF = Moving reference frame
RD = Accuracy in percentage of reading

REFERENCES


Copyright© 2019 by Turbomachinery Laboratory, Texas A&M Engineering Experiment Station
ACKNOWLEDGEMENTS

The authors gratefully acknowledge the financial support of this project by the Verband Deutscher Maschinen- und Anlagenbau e.V. (VDMA), as part of a collaboration with Professor R. Skoda from Ruhr University in Bochum. The Ph.D. work of Mr. Mansour was partially supported by a scholarship from the Egyptian government.