EFFECT OF EXIT BLADE ANGLE, VISCOSITY AND ROUGHNESS IN CENTRIFUGAL PUMPS INVESTIGATED BY CFD COMPUTATION

WEN-GUANG LI

Department of Fluid Machinery, Lanzhou University of Technology, 287 Langongping Road, 730050 Lanzhou, China
liwg43@163.com

(Received 28 January 2011; revised manuscript received 1 March 2011)

Abstract: It has long been known that the exit blade angle plays a very important role in the performance of a centrifugal pump handling water or viscous oil. The effect of exit blade angle on the performance and flow of centrifugal pumps is usually investigated experimentally. However, due to the high cost and limited data that can be obtained by experiment, currently there is a great need for this effect to be studied numerically by means of computational fluid dynamics (CFD). At present, extensive comparisons between experiment and simulation regarding the performance of viscous oil and flow in centrifugal pumps with different exit blade angles are not available. Hydraulic performance and flow details in the impeller and the volute of a centrifugal pump with an exit blade angle of 44° were investigated numerically with the CFD code Fluent®, using water and viscous oil as the working fluid, respectively. The effect of exit blade angle was then elucidated by comparing the performance and flow with that of a pump with an exit blade angle of 20°. It was determined that the results for the performance and flow obtained by means of CFD were in qualitative agreement with the observations. The sudden-rising-head effect is dependent on roughness and viscosity. The unstable zone in the theoretical head curve of the impeller at a low flow rate was attributed to the strong reaction of the volute to the flow in the vicinity of the impeller exit. The flow in the impeller with a large exit blade angle was subject to separation near the blade pressure side, however, a large exit blade angle helped improve the pump performance of viscous oil.

Keywords: centrifugal pump, impeller, volute, viscosity, performance, blade angle, viscous oil

1. Introduction

The effect of exit blade angle on the performance of centrifugal pumps is usually investigated experimentally – for instance [1] for water, [2–9] for viscous oils. These contributions allowed to establish the effect of exit blade angle on the performance of centrifugal pumps. It was shown that in the case of a pump
W. G. Li

handling viscous oils, the exit blade angle has a substantial influence on both the head and efficiency and a large exit blade angle helps to improve both the head and efficiency for liquids with higher viscosity than water. Since the experiments on the performance of pumps and the surveys of flow are very expensive and yield limited data, it is highly desirable that the effect of exit blade angle could be studied numerically by means of CFD in order to obtain sufficient information about the flow pattern and performance parameters. Recently, Shojaee Fard and Boyaghchi [10] have applied CFD codes to examine this effect for highly viscous liquids, and Bacharoudis et al. [11], using a different CFD code – for water. These studies have shown that CFD could aid in clarifying the aforementioned effect. However, these studies have ignored several factors, i.e. the roughness of the wet wall, performance variables of the internal pump (such as the hydraulic efficiencies of the impeller and pump, theoretical head of the impeller, slip factor, hydraulic loss coefficients of the impeller and volute etc.). In addition, the detailed flow conditions in the impeller and volute have not been compared with experiment.

In this paper, the CFD code Fluent® served as a tool for investigating the hydraulic performance and flow of water and viscous oils (of three different viscosities) when these working liquids were transported by a centrifugal pump with an exit blade angle of 44°. The estimated hydraulic performance variables were compared with those obtained for a pump with an exit blade angle of 20°. The presence of the sudden-rising-head effect was confirmed, and the reason for the onset of the unstable theoretical head curve was clarified. A separated flow structure in the impeller with a large exit blade angle was identified.

2. Computational models

2.1. Pump model

The experimental pump of Li [8] was used as the computational model. This was a single-stage, end-suction, overhung centrifugal pump with one suction pipe and a volute. Table 1 shows the specifications of the pump and the primary geometrical dimensions of impellers A and B and of the volute. The impellers have identical design specifications and meridian shape, however, their blade patterns are different. Figure 1 presents a comparison between the blade pattern of impeller A (β₂b = 20°) and impeller B (β₂b = 44°). Unless otherwise stated, the results presented in this paper apply to the pump with impeller B.

Three fluid domains coexist in the computational model. The first one is the fluid in the suction pipe, the second – the fluid in the rotating impeller, and the third one – the fluid in the stationary volute. The interaction of the flow between the impeller and volute was approximately dealt with by the multiple reference frame (MRF) model with averaged unsteady flow. In MRF, the relative position between the impeller and volute is kept constant throughout the computation in order to reduce the computational effort. Such a position was recommended for the geometric modeling of the flow domain and mesh generation processes in
Table 1. Specifications for the pump and the geometry of the impeller and volute

<table>
<thead>
<tr>
<th>Specifications</th>
<th>Impeller A &amp; B</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flow rate, $Q$ (m$^3$/h or L/s)</td>
<td>25 or 6.94</td>
</tr>
<tr>
<td>Head, $H$ (m)</td>
<td>8</td>
</tr>
<tr>
<td>Rotating speed, $n$ (r/min)</td>
<td>1450</td>
</tr>
<tr>
<td>Specific speed, $n_s$ (r/min, m$^3$/s, m)</td>
<td>93</td>
</tr>
</tbody>
</table>

\[ n_s = \frac{3.65n}{Q^{0.75}H^{0.25}} \]

<table>
<thead>
<tr>
<th>Impeller geometry</th>
<th>Impeller A</th>
<th>Impeller B</th>
</tr>
</thead>
<tbody>
<tr>
<td>Eye diameter, $d_e$ (mm)</td>
<td>62</td>
<td>62</td>
</tr>
<tr>
<td>Outlet diameter, $d_2$ (mm)</td>
<td>180</td>
<td>180</td>
</tr>
<tr>
<td>Outlet width, $b_2$ (mm)</td>
<td>18</td>
<td>18</td>
</tr>
<tr>
<td>Blade exit angle, $\beta_{2b}$ (°)</td>
<td>20</td>
<td>44</td>
</tr>
<tr>
<td>Wrapping angle, $\phi$ (°)</td>
<td>120</td>
<td>90</td>
</tr>
<tr>
<td>Number of blades, $Z$</td>
<td>4</td>
<td>4</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Volute geometry</th>
<th>Impeller A &amp; B</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter of base circle, $d_3$ (mm)</td>
<td>190</td>
</tr>
<tr>
<td>Width, $b_3$ (mm)</td>
<td>40</td>
</tr>
<tr>
<td>Area of throat, $F_{th}$ (cm$^2$)</td>
<td>1440</td>
</tr>
<tr>
<td>Diameter of discharge nozzle, $d_9$ (mm)</td>
<td>50</td>
</tr>
</tbody>
</table>

Figure 1. Flow domains (a) and comparison (b) of the shape of the blade in the impellers with a small (A) and large (B) exit blade angle

Gambit. In such a case, the angular rotation speed is still involved in the equations governing the flow (see Fluent 6.3 User’s Guide [12]). The fluid flow in the space between the volute casing, impeller shroud and the hub were not considered. Two sliding mesh interfaces were established between the suction pipe and impeller and between the impeller and volute, respectively. One was located at the impeller
entrance, and the other one – at its outlet. In [13], it was confirmed that the position of the blade relative to the volute tongue influences, to a certain extent, the hydraulic performance. However, the performance corresponding to a position, where the tongue is located exactly in the middle between two successive blades (see Figure 1a), can reasonably approximate the mean performance of the pump. The performance for such a position was considered in the article.

2.2. Physical properties of liquids

Fresh water, oil 1, oil 2 and oil 3 were used as the working liquid, respectively. Their density and kinetic viscosity at 20°C are illustrated in Table 2. It has been confirmed experimentally [8] that these fluids are Newtonian at this temperature.

Table 2. Physical properties of working liquids

<table>
<thead>
<tr>
<th>Liquid</th>
<th>water</th>
<th>oil 1</th>
<th>oil 2</th>
<th>oil 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density, ( \rho ) (kg/m³)</td>
<td>1000</td>
<td>839</td>
<td>851</td>
<td>858</td>
</tr>
<tr>
<td>Kinetic viscosity, ( \nu ) (mm²/s)</td>
<td>1.0</td>
<td>24.47</td>
<td>48.48</td>
<td>60.7</td>
</tr>
<tr>
<td>Nominal viscosity (mm²/s)</td>
<td>1</td>
<td>24</td>
<td>48</td>
<td>60</td>
</tr>
</tbody>
</table>

2.3. Working conditions and flow model

Thirteen working conditions were specified to cover the flow rate range of 1.0–11.0L/s for the pump in the numerical computations. In the pump, the liquid was assumed to be incompressible and its flow was three-dimensional and turbulent, however, the time-averaged flow was steady, which was governed by the continuity equation and Navier-Stokes equations. In the computations, the standard \( k-\varepsilon \) model was employed to evaluate the turbulent stresses. The non-equilibrium wall function was applied to include the shear stress on the walls of the pump in order to account for the effect of the pressure gradient in the primary flow direction on the shear stress as follows:

\[
\begin{align*}
\frac{\partial C_1^{\frac{4}{3}} k^{\frac{3}{2}}}{\partial x} &= \frac{1}{\kappa} \ln \left( E \frac{C_1^{\frac{4}{3}} k^{\frac{3}{2}} y}{\mu} \right) \\
\frac{\partial}{\partial x} W &= W - \frac{1}{2} \frac{\partial p}{\partial x} \left[ \frac{y}{y_0} \ln \left( \frac{y}{y_0} \right) + \frac{y-y_0}{y_0} + \frac{y^4}{2y_0^4} \right]
\end{align*}
\]

where \( W \) is the fluid velocity in the primary flow direction; \( \rho \) and \( \mu \) are the density and the kinematic viscosity of the fluid, respectively; \( \kappa, C_\mu \) and \( E \) are the turbulence constants; \( y_0 \) is the physical viscous sublayer thickness; \( y \) is the normal distance between the wall and the nearest mesh node; \( k \) is the kinetic energy of turbulence; and \( \frac{\partial p}{\partial x} \) is the pressure gradient in the primary flow direction. It can be observed that the effect of \( \frac{\partial p}{\partial x} \) on shear stress \( \tau_w \) is involved in Equation (1). Further information concerning this can be found in Fluent 6.3 User’s Guide [12].

2.4. Numerical schemes

In the computations, a well-established numerical scheme for viscous incompressible flow, SIMPLE, was adopted to solve the steady, three-dimensional,
incompressible and turbulent flow in the pump. An unstructured mesh with quadratic cells was used to separate individual flow domains. It was demonstrated that estimated performance curves no longer change with the mesh size once the number of cells is close to 1030000 (170000 cells in the suction pipe, 330000 cells in the impeller, and 530000 cells in the volute), which was the number of cells used in this paper. In the numerical computations, the under-relaxation factors with respect to pressure, velocities, turbulence kinetic energy and turbulence dissipation rate were set at 0.3, 0.5, 0.8 and 0.8, respectively, with a residual tolerance of $1.5 \cdot 10^{-4}$ in all cases. Under such conditions, the difference in mass flow rate between the inlet and outlet of the pump was insignificant (about $9.72 \cdot 10^{-5} \text{kg/s}$).

2.5. Boundary conditions and wall roughness

The no-slip condition was maintained on the wet walls. At the inlet of the suction pipe, only the uniform axial velocity was present, which could be determined by a given flow rate and the inner diameter of the suction pipe. For major cast surfaces, the roughness is usually $Ra = 12.5–50 \mu m$. In this paper, we assumed that the roughness was $Ra = 0$, 50, 100 $\mu m$, respectively. The corresponding sand roughness was $k_s = 6Ra = 0$, 300, 600 $\mu m$, respectively. The roughness coefficient in Fluent® was set at $C_{ks} = 0.75$.

2.6. Data reduction

The first step consisted in defining several important variables for describing the hydraulic performance of the pump and impeller. The theoretical head of the impeller $H_{th}$ is defined as the total head rise across the impeller. It can be estimated using CFD results. The torque $M_n$ acting on the blades, shroud and the hub of the impeller can be inferred from the CFD results. Subsequently, the shaft power $M_n \omega$ can be determined, where $\omega$ is the rotational angular speed of the impeller and $\omega = \frac{\pi n}{30}$. On the basis of the shaft power, the ideal theoretical head rise $H_i$ can be determined by the following equation:

$$H_i = \frac{\eta_V M_n \omega}{\rho g Q}$$

where the volumetric efficiency of the pump $\eta_V = 0.875$ was estimated based on the experiments in [8]. The hydraulic efficiency of the impeller $\eta_h$ is the ratio of the theoretical head of the impeller to the ideal theoretical head rise: $\eta_h = \frac{H_{th}}{H_i}$. The pump head $H$ is the total head rise across the pump. The difference $\Delta h_i = H_i - H_{th}$ is the hydraulic loss across the impeller, while the difference $\Delta h_V = H_{th} - H$ is the hydraulic loss in the volute. These losses were normalized with the parameter $\frac{u_2^2}{2g}$, yielding the following hydraulic loss coefficients of the impeller and volute $\xi_i = \frac{2\Delta h_i}{u_2^2}$ and $\xi_V = \frac{2\Delta h_V}{u_2^2}$, respectively, where $u_2$ is the speed of the impeller tip and $u_2 = \frac{\omega d}{2}$. Finally, $\xi_i + \xi_V$ is the total hydraulic loss coefficient.

Regardless of the pre-whirl of the fluid at the inlet of the impeller, the theoretical head of the impeller $H_{th}$ is related to the slip factor $\sigma$ and to the
circumferential component of absolute velocity $V_{u2}$ at the outlet of the impeller through the Euler equation. If the mean streamline method is used, as in [14], the theoretical head of the impeller for the one-dimensional (1D) ideal fluid flow can be expressed in terms of the flow rate as:

$$H_{th1D} = \frac{u_2}{g} \left[ (1 - \sigma)u_2 - \frac{Q}{\eta_v A_2 \psi_2 \tan \beta_2 b} \right]$$

where $A_2$ is the area of the impeller exit and $A_2 = \pi d_2 b_2$; $\psi_2$ is the blade lockage coefficient and $\psi_2 = 1 - \frac{Z S u_2}{\pi d_2}$, where $Z$ is the number of blades, $S$ is the circumferential thickness of the blade at the impeller exit and $S_2 = 10\text{mm}$ (impeller A) and $5\text{mm}$ (impeller B). The slip factor $\sigma$ is the ratio of the slip velocity $\Delta V_{u2}$ at the outlet of the impeller to the speed of the impeller tip $u_2$, expressed as follows:

$$\sigma = \frac{\Delta V_{u2}}{u_2}$$

Stodola and Wiesner have proposed the following formulas for the slip factor, see [14]:

$$\sigma = \begin{cases} \frac{\pi}{Z} \sin \beta_2 b \\
\sqrt[2]{\sin \beta_2 b} Z \end{cases}$$

Of course, the slip factor can be determined from Equation (3) using the slip velocity $\Delta V_{u2}$ (see Figure 1b) determined from the CFD results for the three-dimensional (3D) viscous fluid flow. However, the slip factor calculated in this way does not seem to correlate with the theoretical head of the impeller, because the variable $\frac{u_2 V_{u2}}{g}$ no longer exactly represents the total head of viscous fluid flow in the centrifugal impeller. According to Gulich [15], the variable $\frac{u_2 V_{u2}}{g}$ along a streamline of viscous fluid flow can be expressed as follows:

$$\frac{V_{u2} u_2}{g} - \frac{V_{u1} u_1}{g} = \left[ \left( \frac{p_2}{\rho g} + \frac{V_2^2}{2g} \right) - \left( \frac{p_1}{\rho g} + \frac{V_1^2}{2g} \right) \right] + \Delta h_i$$

where $p_1$ and $V_1$ are the static pressure and absolute velocity of fluid at the entrance to the impeller, respectively; $V_{u1}$ is the circumferential component of $V_1$; $V_2$ is the absolute velocity of the fluid at the exit of the impeller; $\Delta h_i$ is the hydraulic loss of the fluid on the streamline from the entrance. In the case without pre-whirl, Equation (6) can be simplified to:

$$\frac{V_{u2} u_2}{g} = \left[ \left( \frac{p_2}{\rho g} + \frac{V_2^2}{2g} \right) - \left( \frac{p_1}{\rho g} + \frac{V_1^2}{2g} \right) \right] + \Delta h_i$$

where the term in the square brackets is the head rise across the impeller, which can be obtained either experimentally or by means of CFD. For an ideal fluid flow, $\Delta h_i = 0$ and $\frac{u_2 V_{u2}}{g}$ is exactly equal to the head rise. For viscous fluid flow, however, $\Delta h_i > 0$ and $\frac{u_2 V_{u2}}{g}$ represents the head rise only approximately.

For the slip factor to be related to the theoretical head of the impeller, it must be estimated from Equation (3), where the 1D theoretical head $H_{th1D}$ is
replaced with the theoretical head of the impeller $H_{th}$ for the 3D viscous fluid flow obtained from CFD:

$$\sigma = 1 - \left( \frac{gH_{th}}{u_2^2} + \frac{Q}{\eta_\nu u_2 A_2 \psi_2 \tan \beta_2} \right)$$  \hspace{1cm} (8)$$

From the CFD computations it was possible to obtain the averaged shear stress applied to wet wall surfaces of the pump. The averaged skin friction factor of the impeller $f_i = \frac{2\tau_{wi}}{\rho u_2^2}$ is defined as the ratio of the averaged shear stress $\tau_{wi}$ in the flow passages of the impeller to the quantity $\frac{cu_2^2}{2}$. Similarly, the averaged skin friction factor of the volute $f_V = \frac{2\tau_{wV}}{\rho u_2^2}$ is defined as the ratio of the averaged shear stress $\tau_{wV}$ in the flow passage of the volute to the quantity $\frac{cu_2^2}{2}$. Both averaged skin friction factors account for the effect of the roughness of the wet walls and for the viscosity on skin friction loss.

In subsequent sections, both the effect of liquid viscosity and of the exit blade angle on these variables will be presented in detail. In particular, the sudden-rising-head effect and the unstable onset in the impeller theoretical head curves will be clarified.

3. Results

3.1. Hydraulic performance of the impeller

Figure 2 shows the theoretical head of the impeller $H_{th}$, the hydraulic efficiency of the impeller $\eta_{hi}$, the hydraulic loss coefficient $\xi_i$, and the slip factor $\sigma$ in terms of the flow rate $Q$ for water, oil 1, oil 2, and oil 3, for a roughness of $Ra=50\mu m$. For comparison, the slip factors obtained by Wiesner and Stodola, respectively, were also illustrated. The 1D theoretical heads estimated from these factors by means of Equation (3) were plotted as well. Note that the theoretical head of the impeller for the 3D viscous fluid flow is quite different from that for the 1D ideal fluid flow. The theoretical head rise of viscous fluid no longer increases steadily with a decrease in flow rate, but, instead, a peak emerges at $Q=6L/s$. The theoretical head of the impeller for the 3D viscous fluid appears to be less sensitive to viscosity.

The agreement between the impeller theoretical head of 3D viscous fluid flow and the corresponding value estimated using the slip factor is better when Wiesner’s and not Stodola’s slip factor is used. This implies that the slip factor proposed by Wiesner is more suitable for the impellers of centrifugal pumps with a lower number of blades and a larger exit blade angle. Since the theoretical head of the impeller for viscous fluid no longer shows a linear dependence on the flow rate, the slip factor extracted from the CFD results is bound to change with the flow rate. Clearly, larger slip factors were obtained at high and low flow rates.

At around $Q=7L/s$, the hydraulic efficiency of nearly 90% was obtained. The vortices of significant intensity in the impeller, an effect of a higher positive angle of attack, lead to a rapid increase in the hydraulic loss coefficient of the impeller at a low flow rate (see Figure 12). For instance, the loss coefficient at
Figure 2. Theoretical head of the impeller (a), slip factor (b), hydraulic efficiency (c) and hydraulic loss coefficient (d) in terms of the flow rate at $Ra=50\mu m$.

$Q = 1\, L/s$ is 25 times larger than that at $Q = 7\, L/s$. When the flow rate exceeds $7\, L/s$, only a slight increase in the loss coefficient is observed.

Both the hydraulic efficiency and the loss coefficient of the impeller distinctly depend on viscosity. Usually, the hydraulic efficiency decreases with an increase in the liquid viscosity. However, when the flow rate exceeds $7\, L/s$, the efficiency at $\nu = 24\, mm^2/s$ (oil 1) is better than at $\nu = 1\, mm^2/s$ (water), leading to improved performance.

3.2. Hydraulic performance of the pump

Figure 3 presents the head of the pump $H$, hydraulic efficiency $\eta_h$, hydraulic loss coefficient of the volute $\xi_V$ and the total hydraulic loss coefficient $\xi_V + \xi_i$ as a function of the flow rate for a roughness of $Ra = 50\mu m$. In most cases, regardless of the flow rate, the head decreases with an increase in viscosity. At the viscosity of $24\, mm^2/s$, the head is the highest and the sudden-rising-head effect occurs. At a low flow rate of $4\, L/s$, a negative slope emerges in the head curves resulting in
an unstable head curve, which may be associated with the stagnation of the flow in the impeller.

The pump efficiency $\eta$ was obtained numerically from the hydraulic efficiency of the pump $\eta_h$, calculated by CFD, and the volumetric efficiency $\eta_V$, which was determined from the experimental data in [8]. It was shown that the estimated efficiency of the pump is usually 2–3% lower than the experimental value, except in the case of liquid with the viscosity of $\nu = 24\text{mm}^2/\text{s}$. At this viscosity, the most desirable head and efficiency of the pump can be obtained at various flow rates. This phenomenon is termed the sudden-rising-head effect.

The best estimated hydraulic efficiency of the pump was 75%, whereas the best hydraulic efficiency of the impeller was 90%. In the volute, the hydraulic loss increases by at least 15%. In comparison with Figure 2, panel (c), in the impeller, the hydraulic loss is only 10%. The volute plays a very important role in the determination of hydraulic loss in the pump.

The difference in the coefficient curves for hydraulic loss of the volute and the impeller is distinct. The loss coefficient of the volute increases with an increase in the flow rate, whereas the loss coefficient of the impeller decreases significantly until $Q = 7\text{L}/\text{s}$. When the flow rate was increased from 1L/s to 11L/s, the increase in the loss coefficient of the volute was 80%.

The minimum total hydraulic loss coefficient of the pump was obtained near $Q = 7\text{L}/\text{s}$. At a flow rate below $Q = 7\text{L}/\text{s}$, the total loss coefficient increases rapidly with a decrease in the flow rate. However, at a higher flow rate, the coefficient increases only slightly with an increase in flow rate. For instance, at $Q = 7\text{L}/\text{s}$, the hydraulic loss coefficient of the volute is 80% larger than that of the impeller; whereas at $Q = 11\text{L}/\text{s}$, these coefficients are identical. At low flow rates, the hydraulic loss in the impeller is substantial, whereas at high flow rates the hydraulic loss in the volute is significant. Therefore, in the case of pumps with larger exit blade angles, the reduction of the hydraulic loss in the volute can help improve their performance.

### 3.3. Sudden-rising-head effect

It was shown in Figure 3 that regardless of the flow rate, the head rise and the hydraulic efficiency of the pump decrease continuously with an increase in viscosity. However, the head and hydraulic efficiency are higher at $\nu = 24\text{mm}^2/\text{s}$ (oil 1) than at $\nu = 1\text{mm}^2/\text{s}$ (water), which causes the sudden-rising-head effect, see [8]. It was observed that the sudden-rising-head effect was taking place when the viscosity of the liquid was in the range of 24–60mm$^2$/s, see [8]. Note that the sudden-rising-head effect obtained by CFD computation seems maintainable at $\nu = 24\text{mm}^2/\text{s}$. The CFD results were in partial agreement with experiment.

In [13], for a roughness of $Ra = 0\mu\text{m}$, the head curves obtained by CFD were successively decreasing with an increase in viscosity, and the sudden-rising-head effect did not occur. However, once the roughness was set at $Ra = 100\mu\text{m}$, the effect was again observable. This demonstrates that the sudden-rising-head effect is dependent on roughness and friction.
Figure 3. Head of the pump (a), efficiency (b), hydraulic efficiency (c), hydraulic loss coefficient of the volute (d) and total hydraulic loss coefficient (e) in terms of the flow rate at Ra = 50µm
Figure 4 presents the average skin friction factors for the wet surface of
the impeller and volute, respectively, at the roughness of Ra = 50µm. Their
magnitude is similar, however, their dependence on the flow rate is somewhat
different. The skin factor of the impeller surface assumes minimum values for the
flow rate in the range of 5–7L/s, whereas the skin factor of the volute surface
increases with an increase in flow rate. Usually, a higher viscosity results in a larger
skin friction factor in the impeller and volute. However, the lowest skin friction
factor in the impeller was obtained for a viscosity of 24mm²/s, regardless of the
flow rate. The lowest skin friction factor in the volute, for this viscosity, was
observed when the flow rate exceeded 2.5L/s. A decrease in the skin friction
factor implies a reduction in skin friction loss. In this case, the performance of the
pump improved, which is demonstrated by a higher head and hydraulic efficiency.
In consequence, the sudden-rising-head effect occurred in the impeller and in the
volute due to a decline in the skin friction loss.

![Figure 4](image)

Figure 4. Averaged skin friction coefficients of the impeller (a) and volute (b)
as a function of the flow rate at Ra=50µm

The sudden-rising-head effect can be interpreted by means of the relation
between the skin friction factor and the Reynolds number of the fluid flow in
circular pipes or boundary layer flow over a flat plate. Figure 5 presents the
curves of the skin friction factor $f$ against the Reynolds number $Re$ in circular
pipes, according to White [16]. For a constant flow rate, a smaller skin friction
factor means a lower skin friction loss in the pump, impeller or volute. This may
cause the sudden-rising-head effect, when the Reynolds number is decreased due
to an increase in the viscosity of the fluid. Figure 5 shows that in a smooth
pipe, there is a zone I between the turbulent and laminar regimes, where the skin
friction factor is relatively small. At low roughness, in the transitional regime,
there is a zone II between the fully rough and smooth regimes. In zone II,
a smaller skin friction factor was obtained – such an effect is known as inflectional
behavior [17]. At high roughness, there is a zone III between the fully rough and laminar regimes. In zone III, it was possible to obtain a significantly smaller skin friction factor than in the fully rough regime. In small centrifugal pumps, the relative roughness (the ratio of hydraulic diameter to roughness) is also small; the flow in the impeller and volute is in the fully rough regime at high Reynolds numbers (lower viscosity). When the Reynolds number decreases with an increase in viscosity, the skin friction factor also decreases in zone III, causing the sudden-rising-head effect. In large pumps, the zone with a smaller skin friction factor can occur in the transitional regime, and therefore, the sudden-rising-head effect may persist there. The zones with a lower skin friction factor can also be found in the plots of the skin friction factor of boundary layer flow over flat plates against the Reynolds number [16]. Thus, in the case of flat plates, the sudden-rising-head effect can be explained in the same way by means of the skin the friction factor.

![Figure 5. Skin friction factor against Reynolds number for circular pipes](image)

3.4. *Unstable zone in the theoretical head curves*

Figure 6 illustrates the static pressure rise and an increase in the kinetic head of the liquids across impeller B with respect to the flow rate. It was shown that regardless of the viscosity, a steady growth occurs in the static head rise with a decrease in the flow rate. However, the kinetic head rise displays a peak at a flow rate of about 5L/s and subsides rapidly at both ends of the curve obtained for this flow rate. A considerable decline in the kinetic head rise at a low flow rate is responsible for the appearance of the unstable zone in the theoretical head curves, see Figure 2a. Such a low kinetic head rise was attributed to a significant decline in the circumferential component of absolute velocity at a low flow rate.
Effect of Exit Blade Angle, Viscosity and Roughness in Centrifugal Pumps...

Figure 6. Static head rise (a) and the kinetic head rise (b) of liquids across the impeller in terms of the flow rate at Ra = 50µm

Figure 7. Circumferential (a) and radial (b) components of absolute velocity of fluids at different viscosities at the discharge of the impeller in terms of the flow rate.

Figure 7 presents the radial and circumferential components of absolute velocity of a liquid at the discharge of the impeller with respect to the flow rate. The two components of the 1D ideal fluid flow at the discharge were estimated using ideal 1D fluid flow theory (for centrifugal pumps) and Wiesner’s slip factor correction [14]. Unlike the circumferential component of absolute velocity in 3D viscous liquids, the circumferential components of the 1D ideal fluid flow rise steadily with a decrease in the flow rate. Since the variable $\frac{V_\theta u_2}{g}$ stands for the total head of the ideal fluid at the outlet of the impeller, such a variation of $V_\theta$ implies that the 1D theoretical head of the impeller in the centrifugal pump increases with a decrease in the flow rate. For viscous fluids, however, the variable $\frac{V_\theta u_2}{g}$ does not exactly stand for the total head at the outlet of the impeller. Unfortunately, this leads to a decrease in the circumferential velocity of...
the 3D flow in viscous fluids with a decrease in the flow rate. Although the radial component of absolute velocity, which increases with the flow rate, is shown for both inviscid and viscous liquids, the radial velocity for the 3D viscous flow is about 100% higher than that of the 1D inviscid flow, because of the blockage caused by the boundary layers and the non-uniform velocity profile of the flow passages of the impeller.

The volute of a centrifugal pump is situated right behind the impeller (the radial gap between the volute tongue, or cutwater, and the outlet of the impeller is as small as 1.5–2.5% of the impeller diameter), and therefore the liquid in the volute can have a strong impact on the fluid at the impeller discharge. The averaged circumferential velocity of the liquid in the volute is proportional to its flow rate [14], while the circumferential velocity at the discharge of the impeller increases with a decrease in flow rate. Thus, at a high flow rate, the fluid in the volute will push the nearby fluid towards the outlet of the impeller. At a low flow rate, however, the liquid in the volute will pull the nearby fluid from the outlet of the impeller. This effect is responsible for a significant decrease of the circumferential velocity of the liquid at the outlet of the impeller at a low flow rate. The velocity profiles observed in the volute in [8], confirm the existence of this effect.

When, instead of the volute, a vaneless diffuser was fitted behind the centrifugal impeller, as in [18], the circumferential velocity increased with a decrease in the flow rate. In this case, the flow at the discharge of the impeller was not affected by the vaneless diffuser.

### 3.5. Effects of exit blade angle on performance

Figure 8 illustrates the pump head $H$, efficiency $\eta$, average skin friction factors of the impeller $f_i$, and volute $f_V$ in terms of the flow rate at $Ra=50\mu\text{m}$ and $\nu=1\text{mm}^2/\text{s}$, and $\nu=60\text{mm}^2/\text{s}$ for the pump with impeller A (exit blade angle of 20°) and impeller B (exit blade angle of 44°) in order to make a comparison between the experiment [8] and simulation (this paper). For impeller A, the head curves calculated by CFD show good agreement with experiment (less than 10% error), however, for impeller B, the agreement was less than satisfactory. In the experiments, the sudden-rising-head effect occurred when the viscosity was in the range of 24–60mm$^2$/s, however, in CFD calculations, this effect occurs only at $\nu=24\text{mm}^2/\text{s}$.

At the viscosity of $\nu=1\text{mm}^2/\text{s}$, impeller A demonstrates the highest efficiency, higher than the efficiency of impeller B at a flow rate of $Q=6\text{L/s}$. Unfortunately, when the viscosity was increased to 60mm$^2$/s, the efficiency of impeller A was even lower than that of impeller B. Moreover, the higher the flow rate, the lower the efficiency. These numerical results show very good agreement with the experiment carried out by Li [8]. Both CFD and experiment have shown that the highest efficiency can be achieved at a higher flow rate as the blade angle increases.
3.6. Effect of exit blade angle on the flow

Figure 9 shows a comparison between the relative velocity vectors calculated by CFD at a radius of 50mm, 70mm and 85mm, respectively, in the mid-span plane of impeller A and impeller B at $Q = 6\, \text{L/s}$ (the point of best efficiency). The average skin friction factor in the volute for impeller B is higher than that in the volute for impeller A, regardless of the flow rate. This indicates that a large exit blade angle induces additional hydraulic loss in the volute. Fortunately, when the flow rate exceeds 6L/s, the skin friction factor of impeller B is considerably reduced, which improves the performance of the entire pump.

However, the average skin friction factor of impeller B is substantially higher than that of impeller A at a flow rate lower than 6L/s. This effect is responsible for deteriorated efficiency of the pump with impeller B at low flow rates.

Figure 8. Head of the pump (a), efficiency (b), averaged impeller (c) and volute (d) skin friction coefficients in terms of the flow rate for the pumps with a small (A) and large (B) exit blade angle of the impeller.
Figure 9. Relative velocity vectors at a radius of 50, 70 and 85mm, respectively, in the mid-span plane of the impeller at $Q = 6\text{L/s}$ and $\text{Ra} = 50\mu\text{m}$. The CFD results are on the left-hand side and the LDV measurements are on the right-hand side. Panels (a)–(d) correspond to impeller A; (e)–(h) correspond to impeller B. Panels (a), (b), (e) and (f) correspond to a viscosity of $1\text{mm}^2/\text{s}$; (c), (d), (g) and (h) correspond to a viscosity of $48\text{mm}^2/\text{s}$.
and $Ra = 50 \mu m$ at a viscosity of $1 \text{mm}^2/\text{s}$ and $48 \text{mm}^2/\text{s}$ and the Laser Doppler velocimetry (LDV) measurements in [8]. The flow in impeller A was not subject to separation, regardless of viscosity, in both CFD and LDV. However, the flow in impeller B was subject to a significant separation near the pressure side of the blade in both CFD and LDV. A small difference in the relative velocity profiles of the two blades was identified at different viscosities. With CFD, no separation was observed near the suction side of the blade in either of the impellers. In LDV, however, the potential flow core shifted from the suction side of the blade (at low viscosity) to the middle of the channel (at high viscosity) due to the effect of high viscosity. The CFD approach failed to explain this change in the velocity profile. This implies that the velocity profile obtained by Fluent® was insensitive to the viscosity of the working fluid. It was reported that the physical viscosity of the fluid was ignored in the case of fully developed turbulent flows when the standard $k$-$\varepsilon$ turbulence model was applied, see the Fluent 6.3 Manual [12].

Since the cross section VIII (see Figure 10) in the volute plays an important role in the performance and hydraulic design of the pump, as shown by Gulich [15] and Kurokawa and Hode [18], the circumferential velocity of the fluid was examined in this section. Figure 11 illustrates the circumferential velocity profiles along the four lines in section VIII for impeller A and impeller B at a viscosity of $1 \text{mm}^2/\text{s}$ and $48 \text{mm}^2/\text{s}$, respectively, and at $Q = 6 \text{L/s}$ and $Ra = 50 \mu m$. Those lines differ in the distance to the discharge of the impeller and are in the transverse (span) direction of the volute. A more uniform circumferential velocity profile occurs with an increase in the distance from the discharge of the impeller, in which case there is an agreement between CFD and LDV. However, several local details of the profile near the impeller show substantial differences between the CFD results and LDV measurements, particularly for impeller B. CFD calculations confirmed that the more viscous the fluid, the higher its circumferential velocity. This was also indicated by LDV experiments, particularly for impeller A.

![Figure 10. Four cross-sections of the volute (a) and the local coordinate $x, y$ in the section VIII (b)](image)

Effect of Exit Blade Angle, Viscosity and Roughness in Centrifugal Pumps...
Figure 11. Circumferential velocity profile across the volute at four cross-sections with a different distance from the outlet of the impeller in section VIII at $Q = 6L/s$ and $Ra = 50\mu m$. The CFD results are on the left-hand side and the LDV measurements are on the right-hand side. Panels (a)–(d) correspond to impeller A; (e)–(h) correspond to impeller B. Panels (a), (b), (e) and (f) correspond to a viscosity of $1mm^2/s$; (c), (d), (g) and (h) correspond to a viscosity of $48mm^2/s$

Figure 12. Relative velocity vectors obtained by CFD in the mid-span plane of impeller A and impeller B at $Q = 6L/s$ and $Ra = 50\mu m$. Panels (a) and (b) correspond to impeller A; (c) and (d) correspond to impeller B. Panels (a) and (c) correspond to a viscosity of $1mm^2/s$; (b) and (d) correspond to a viscosity of $48mm^2/s$
Figure 12 shows that the separated flow on the pressure side of the blade in impeller B leads to a discharge of the flow from the impeller into the volute at a small relative flow angle, which causes large slip effect. This effect is responsible for the abnormal circumferential velocity profile near the outlet of impeller B, as shown in Figure 7.

Since impeller B is subject to a large exit blade angle, separated flow occurs in the channels of the impeller (see Figure 12), which is the reason behind the relatively poor agreement of the flow details between LDV and CFD. Thus, further research is needed and there is a great demand for more advanced turbulence models, cf. [19] and [20]. The effect of the turbulence model on the flow and performance of impellers with large exit blade angles will be addressed in the near future. Furthermore, the unsteady flow in the impeller is also worth investigating.

4. Conclusions

The hydraulic performance of a centrifugal pump with a larger exit blade angle was studied numerically with the CFD code Fluent® for the pump handling water and different viscous oils. The effect of the viscosity of the working fluid on the performance of the pump was identified. The dependence of viscosity on the theoretical head of the impeller, slip factor, hydraulic efficiency and hydraulic loss coefficient was examined quantitatively. The reasons for the sudden-rising-head effect and the onset of the unstable curve in the theoretical head curves of the impeller at a low flow rate were clarified. It was confirmed that the CFD results were in qualitative agreement with experiment and thus that CFD is able to handle the effect of the exit blade angle on the performance and flow of a pump, regardless of viscosity. It was shown that a large exit blade angle results in an increase in the hydraulic loss over the entire flow rate range in the volute. However, it caused an increase in hydraulic loss at a low flow rate and reduced the loss at a high flow rate in the impeller. The flow in the impeller with a large exit blade angle acquires a separated structure near the pressure side of the blade even at the point of best efficiency. Elucidating the evolution of such a flow pattern in the impeller by means of more advanced turbulence flow models at various working conditions is a challenge for hydrodynamics researchers.

References

Effect of Exit Blade Angle, Viscosity and Roughness in Centrifugal Pumps...
