Article

CFD and Energy Loss Model Analysis of High-Speed Centrifugal Pump with Low Specific Speed

Rong Xue *, Xinyi Lin , Beile Zhang, Hong Zhou, Tianwei Lai and Yu Hou

School of Energy and Power Engineering, Xi’an Jiaotong University, Xi’an 710049, China; ytriangle@stu.xjtu.edu.cn (X.L.); zbeile@stu.xjtu.edu.cn (B.Z.); zh20000418@163.com (H.Z.); laitianwei@mail.xjtu.edu.cn (T.L.); yuhou@mail.xjtu.edu.cn (Y.H.)
* Correspondence: xuerong@mail.xjtu.edu.cn

Abstract: High-speed centrifugal pumps with low specific speed have the characteristics of a small flowrate, a high head, and being compact and light weight, making them promising candidates for applications in the thermal management of aerospace and electronic devices. The energy loss in the low specific speed pump is critical and complex due to the large impeller diameter, the narrow and long flow channel, and the small outlet width. In this paper, an analysis method based on an energy loss model and computational fluid dynamics simulations (ELM/CFD) is proposed to analyze the performance of the low specific speed pump with a fully sealed structure. Experiments were carried out under variable water flowrates. The results show that the empirical correlation method failed to accurately predict the performance of high-speed centrifugal pumps, because the bearing clearance leakage and motor channel leakage are ignored. Moreover, the volume loss and hydraulic loss are calculated based on the empirical parameters of commonly used pumps that are different from the high-speed pump with the low specific speed in the complex flow channel structure. The ELM/CFD method calculates various loss power based on the simulation results and can predict the head and efficiency with deviations less than 2% and 5%, respectively. ELM/CFD can accurately analyze the optimization direction of the pump. The hydraulic loss and the volume loss of the impeller are the dominant factors that restrict the pump efficiency under the lower flowrates, while the hydraulic loss of subsequent flow channels becomes important under the larger flowrates.

Keywords: centrifugal pump; low specific speed; energy loss model; CFD simulation

1. Introduction

The pump is the “heart” of modern industry. The centrifugal pump accounts for more than 70% of the market share for its advantages of the stable flow, simple structure, small volume, and low cost [1]. The high-speed centrifugal pump with a specific speed between 30–80 has the characteristics of the high head, small flow, and high speed. It plays an irreplaceable role in the aerospace, biomedicine, petrochemical, transportation, and other industries [2]. However, its characteristics of small flow and high head also lead to the different structure and hydraulic characteristics between the low specific speed pump and the ordinary centrifugal pump. The impeller diameter of the low specific speed pump is larger and the outlet width is smaller, resulting in larger friction loss in the impeller channel. Meanwhile, both the large hydraulic loss and the shock loss are caused by the reflux at the inlet, the secondary flow in the channel, and the jet wake flow at the outlet. The empirical correlations of the ordinary centrifugal pump cannot predict the power loss of the low specific speed pump correctly, resulting in the low efficiency and poor working stability of the low specific speed centrifugal pump that is designed by the empirical correlation method [3,4]. A high-speed satellite propulsion pump designed by NASA has been tested, the results of which are different from the prediction based on the similarity theory of large pumps with a deviation more than 25%, indicating that the design method of the ordinary
centrifugal pump is not applicable to the low specific speed and high-speed centrifugal pump. As a result, the development of the low specific speed pump has critical problems due to the high cost and the long period that includes theoretical design and multiple prototype tests [5].

Many performance prediction methods have been proposed, including the empirical correlation method, computational fluid dynamics (CFD), and the energy loss model (ELM). Choi et al., calculated the performance of various low specific speed pumps by the empirical correlation method, which were compared with the test results. Their study illustrated that the impeller structure has a great influence on the error of pump performance predicted by the empirical correlation method [6,7]. Tan et al., predicted the performance of the low specific speed pump based on the pressure, speed, and torque in the FLUENT numerical simulation results [8–10]. Chabannes et al., used CFX to simulate a low specific speed pump and obtained the predicted performance of the pump in the post-processing [11,12]. Cui et al., conducted three-dimensional turbulent flow numerical simulations of four different low specific speed centrifugal pumps through FINE/TURBO™ 6.2 and predicted the performance of the pump [13]. Yang and Zhang proposed a method to calculate the optimum flow, the head, and the specific speed coefficient of the low specific speed centrifugal pump, and the effectiveness of the theoretical method was verified by experiments [14]. The National Engineering Laboratory (NELB) had successfully applied ELM optimization to the design of centrifugal pumps [15]. Fan et al., investigated the energy loss and performance of the pump by the numerical analysis using the entropy generation method [16,17]. Omar established the energy loss model with the energy loss equation based on the existing theory and the experience, and a program for predicting the performance of centrifugal pump was designed and verified by experiments [18]. Del et al., improved the full Reynolds stress model by CFD to improve the accuracy of performance prediction of the low specific speed pump [19]. Most of the existing numerical calculations can only predict the overall performance without the detailed analysis of various energy losses. Moreover, the energy loss of the low specific speed pump with a fully sealed structure is more complex. The hub leakage, bearing clearance leakage, and motor channel leakage have significant impacts on the pump performance, which are commonly ignored by the empirical correlation method, resulting in the high head and efficiency of the predicted low specific speed pump performance [20]. The ELM method can be applied to various structural pumps for considering the power of each part. However, ELM adopted in the relative literature is based on empirical theoretical parameters that will influence the prediction accuracy and it is less used in the pump performance analysis. Therefore, ELM optimization and CFD optimization were combined to form a new method in this study, namely the ELM/CFD method, which can calculate various energy losses of the low specific speed centrifugal pump based on the numerical simulation and evaluate their respective or joint effects on the pump performance.

In this paper, the experiments were performed on a high-speed centrifugal pump with the low specific speed and the performance curve of the pump under 7500 rpm was obtained. A method combining the energy loss model and CFD simulations was used to reveal the loss mechanisms. The comparison between ELM/CFD predictions and the experimental results shows good agreement, indicating the ELM/CFD method can be a useful tool for the design of high-speed centrifugal pumps with the low specific speed. The results of this paper also provide guidelines for the development of efficient high-speed centrifugal pumps.

2. CFD Simulations and Energy Loss Model
2.1. Structural Parameters of Pump

A new type of oil-free high-speed centrifugal pump with the low specific speed has been developed in our previous studies [21,22]. The design parameters of the centrifugal water pump are flow rate $Q = 3 \text{ m}^3\cdot\text{h}^{-1}$, head $H = 30 \text{ m}$, and rotational speed $n = 7500 \text{ rpm}$. The structural parameters of the hydraulic components are shown in Table 1.
The image of the pump is shown in Figure 1a. Referring to the physical pump, a three-dimensional model of the low specific speed high-speed centrifugal pump was established, and the general assembly drawing of the pump is shown in Figure 1b. The pump consists of dynamic parts and static parts. The dynamic parts contain the inducer, the impeller, the hydrodynamic bearings, and the main shaft, while the rest are the static parts, including the inlet and outlet cover plate, the vaneless diffuser, the pump shell, the motor stator, and other supporting parts. The space between the dynamic and the static parts is the flow channel. As the red line in Figure 1b shows, the liquid flows into the impeller after being pressurized in advance in the inducer (A to B), which improves the cavitation performance. In the impeller, the energy of the motor is transformed into the power of the liquid (B to C), and the energy is transformed into the pressure energy when the liquid flows through the vaneless diffuser (C to D) and the vaned diffuser (D to E). A small stream of liquid flows through the internal flow channel to lubricate the bearing (C to F). The liquid flows through the motor channel cooling motor (E to G), then the liquid flows out of the outlet (G to H).

![Image of the low specific speed high-speed centrifugal pump](a)

**Figure 1.** Image of the low specific speed high-speed centrifugal pump (a) and schematic of low specific speed high-speed centrifugal pump; (b): 1—inducer; 2—impeller; 3—diffuser; 4—casing; 5—main axle; 6—motor stator; 7—tail end fixed bearing; 8—front fixed bearing.

### 2.2. CFD Simulations

As for the water pump studied in this paper, the fluid can be regarded as incompressible fluid, that is, the density is constant, and both the change of temperature and the energy conservation equation can be ignored. Thus, the isothermal model with 25 °C is adopted. The inlet condition of simulation is the total pressure, 0.1 MPa, and the outlet condition is the mass flow rate from 1 to 5 m³·h⁻¹. Meanwhile, the convergence criterion in the Solver Control is set as 10⁻⁵.

As for the incompressible turbulent flow, the Reynolds time averaged method, which is usually adopted in engineering, is used to deal with the transient Navier Stocks equations. The instantaneous value of each pulse momentum in the flow field is expressed as the

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Symbol</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Semi-open impeller</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Number of blades</td>
<td>$Z_1$</td>
<td>6</td>
</tr>
<tr>
<td>Inlet diameter/mm</td>
<td>$D_1$</td>
<td>27.2</td>
</tr>
<tr>
<td>Outlet diameter/mm</td>
<td>$D_2$</td>
<td>64.0</td>
</tr>
<tr>
<td>Outlet blade width/mm</td>
<td>$b_2$</td>
<td>3.8</td>
</tr>
<tr>
<td>Outlet angle/°</td>
<td>$\beta_2$</td>
<td>20</td>
</tr>
<tr>
<td>Vaned diffuser</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Number of blades</td>
<td>$Z_2$</td>
<td>7</td>
</tr>
<tr>
<td>Inlet diameter/mm</td>
<td>$D_3$</td>
<td>65.0</td>
</tr>
<tr>
<td>Outlet diameter/mm</td>
<td>$D_4$</td>
<td>82.0</td>
</tr>
<tr>
<td>Outlet angle/°</td>
<td>$\beta_3$</td>
<td>5.8</td>
</tr>
</tbody>
</table>
sum of the average value and the pulsation value, and the characteristics of turbulence are expressed by other equations. The Navier Stocks equations are shown below.

$$\frac{\partial \tau_{ij}}{\partial x_j} = 0$$  \hspace{1cm} (1)

$$\frac{\partial \rho \tau_{ij}}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho \tau_{ij} u_j \right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \rho \bar{u}_i u_j \right] + f_i$$  \hspace{1cm} (2)

The $k$-$\varepsilon$ model is the most widely used one for its good robustness and reasonable prediction of large-scale turbulence. The dissipation rate is defined as:

$$\varepsilon = \frac{\mu}{\rho} \left( \frac{\partial u_i}{\partial x_i} \right)$$  \hspace{1cm} (3)

Turbulent viscosity $\mu_t$ is a function of $k$ and $\varepsilon$:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$  \hspace{1cm} (4)

Then, there is the standard $k$-$\varepsilon$ turbulence equation for incompressible fluid:

$$\frac{\partial}{\partial t} \left( \rho k \right) + \frac{\partial}{\partial x_i} \left[ \rho \mu_k \left( \frac{\partial u_i}{\partial x_i} \right) \right] = \rho \left( p_k - \varepsilon \right)$$  \hspace{1cm} (5)

$$\frac{\partial}{\partial t} \left( \rho \varepsilon \right) + \frac{\partial}{\partial x_i} \left[ \rho \mu_k \left( \frac{\partial u_i}{\partial x_i} \right) \right] = \rho \varepsilon \left( C_1 p_k - C_2 \varepsilon \right)$$  \hspace{1cm} (6)

where $C_\mu, \sigma_k, \sigma_\varepsilon, C_1, C_2$ refer to the turbulence model coefficient and the values are 0.09, 1.00, 1.30, 1.44, 1.92, respectively.

As for the compressible flow, the pressure correction algorithm SIMPLE method is used to solve the governing equations. The channels of inducer and impeller are set as a rotating state at 7500 rpm, of which the blades, shroud, and hub are set as a rotating wall. The surfaces of the internal flow channel are also set as a rotating wall and the rest of the surfaces are set as a frozen wall. The wall function in ANSYS CFX, which was first proposed by Launder and Spalding, establishes the relationship between relevant variables near the wall and wall conditions. In the logarithmic interval, the relative relationship between the near wall tangent velocity and the wall shear stress can be expressed as:

$$u^+ = \frac{U_t}{\mu_t} = \frac{1}{\kappa} \ln \left( y^+ \right) + C'$$  \hspace{1cm} (7)

where $u$ is velocity near wall, $u_t$ is friction velocity, and $U_t$ is the component of wall tangential velocity at $y$. In the equations above, $y$ is the dimensionless distance of wall, $C'$ and $\kappa$ are constant.

Owing to the fact that various flow channels are greatly different in the structure and size, different software has been used to mesh them. The flow channel of turbomachinery such as the inducer, the impeller, and the vaned diffuser are meshed by TurboGrid, and the First Element Offset method is used with 30–80 Y+. The inlet section, the vaneless diffuser, the motor flow passage, and the outlet section are meshed by ICEM. The hub clearance and the flow passage of the bearings are meshed by GAMBIT. The scale of the tip clearance, the hub clearance, and the liquid film thickness of the bearings are regarded as constants in the simulation. The ordinary interpolation method was adopted for the connection of the ordinary stationary area, and the frozen rotor method was used for the connection between the rotating area and the stationary area. The meshes are shown in Figure 2.
To test the influence of grid density on the results, five cases with different grid numbers are simulated, as shown in Figure 3. With the increase in grid number, the pump head almost keeps unchanged when the grid number is increased beyond eight million.

![Figure 2. The mesh of low specific speed high-speed centrifugal pump.](image)

![Figure 3. Mesh independence study.](image)

2.3. Empirical Correlations for Ordinary Pumps

The external characteristic prediction of the ordinary pump usually adopts empirical correlations and the efficiency of the pump is calculated by Equation (8). The disc friction loss is determined by Equations (9) and (10) [23]. Since the specific speed of the centrifugal pump is lower than 65 under the design condition, the theoretical head is calculated by the Wiesner correlation (Equation (11)).

\[
\eta = \frac{1}{\eta_v \eta_h} + \frac{P_m}{P_w} + 0.03 \tag{8}
\]

\[
P_m = 0.0035k\rho\omega^3 \left( \frac{D_2}{2} \right)^5 \tag{9}
\]

\[
k = 0.027 + 4.75\left( \frac{n_s}{100} \right) - 8.56\left( \frac{n_s}{100} \right)^2 + 6.23\left( \frac{n_s}{100} \right)^3 - 1.61\left( \frac{n_s}{100} \right)^4 \tag{10}
\]

\[
H_t = \frac{\pi D_2 n}{60^2} \left( \frac{\pi D_2 n}{60} - \frac{Q_I}{3600\pi D_2 b_2 \psi_2 \tan \beta_2} \right) \tag{11}
\]

where \(\eta_v\) is volumetric efficiency, \(\eta_h\) is hydraulic efficiency, \(P_m\) is disc friction loss, \(\omega\) is angular velocity, \(\sigma\) is slip coefficient, and \(\psi_2\) is blade outlet extrusion coefficient.
2.4. Energy Loss Model for High-Speed Centrifugal Pumps

The external characteristic analysis model of the high-speed centrifugal pump is established by using the energy loss model (ELM). The total power of the pump is the sum of the mechanical loss power \( P_m \), volume loss power \( P_v \), hydraulic loss power \( P_h \), and useful power \( P_u \), and the mechanical power loss of the low specific speed pump is mainly the disc friction loss.

The volume loss refers to the partial fluid in the centrifugal pump that will not be discharged from the pump due to the existence of the leakage channel, including three parts: the internal flow \( P_{v,ib} \), the tip leakage of inducer \( P_{v,ind} \), and the tip leakage of impeller \( P_{v,imp} \). The volume loss of each part is composed of the input work, the product of leakage, and the differential pressure between the inlet and the outlet, as shown below:

\[
P_v = P_{v,ib} + P_{v,ind} + P_{v,imp} = q_b \Delta p_b + M_b \omega + q_{tip} \Delta p_{tip} + M_{tip} \omega + q_{imp} \Delta p_{imp} + M_{imp} \omega
\]  

(12)

The hydraulic power of the centrifugal pump is calculated by the following formulas:

\[
P_h = P_{h,ind} + P_{h,imp} + P_{h,dif} + P_{h,dif} + P_{h,rot} + P_{h,rot}
\]  

(13)

The actual useful power of the centrifugal pump is calculated by the following formulas:

\[
P_u = \rho g Q H
\]  

(14)

\[
H = \frac{P_{out}}{\rho g} - \frac{P_{in}}{\rho g} + \Delta z
\]  

(15)

The total efficiency \( \eta_{all} \) can be divided into three parts: the mechanical efficiency \( \eta_m \), the volumetric efficiency \( \eta_v \), and the hydraulic efficiency \( \eta_h \). The mechanical efficiency is the ratio of the power minus the friction loss to total power. The volumetric efficiency is the ratio of the theoretical useful work to that without volumetric loss. The hydraulic efficiency represents the ratio of the actual useful power to the theoretical useful power considering the volume loss, as follows:

\[
\eta_{all} = \eta_m \eta_v \eta_h = \left(1 - \frac{P_u}{P_{all}}\right)\left(\frac{P_u + P_h}{P_u + P_v}\right)\left(\frac{P_u}{P_u + P_h}\right) = \frac{P_u}{P_{all}}
\]  

(16)

3. Results and Discussion

3.1. Comparison of Simulation Results with Experimental Data

The open-mechanical test bed was modified from Refs. [18,19], as shown in Figure 4. The system includes the water storage tank, the flow valve, the electric control valve, the turbine flowmeter, the surge tank, the data acquisition system, and the high-speed centrifugal pump. The uncertainty is listed in Table 2.

![Figure 4. Comparison between numerical simulation results and experimental data. (a) Graph of system; (b) image of system.](image-url)
Table 2. Experimental uncertainty.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Uncertainty/%</th>
<th>Measurement Span</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure/MPa</td>
<td>±0.25</td>
<td>0–1.5</td>
</tr>
<tr>
<td>Power/kW</td>
<td>±0.5</td>
<td>0–10</td>
</tr>
<tr>
<td>Flowrate/m³·h⁻¹</td>
<td>±0.5</td>
<td>0.5–10</td>
</tr>
<tr>
<td>Head/m</td>
<td>±0.35</td>
<td></td>
</tr>
<tr>
<td>Efficiency/%</td>
<td>±1.17</td>
<td></td>
</tr>
</tbody>
</table>

The performance test of the centrifugal water pump was carried out at the design speed of 7500 rpm. The flowrate was changed by adjusting the opening of the electric control valve and the speed of the motor. The flowrate, power, head, and efficiency under different water flow conditions were obtained by the data acquisition system, and the results are shown in Table 3. As the flowrate in the experiment increases, the pump head decreases faster and faster, and the efficiency firstly increases and then decreases after reaching 35.9% at \( Q = 4 \text{ m}³·\text{h}^{-1} \). As for the pump with the low specific speed that was designed by the empirical correlation method, the flowrate corresponding to the highest efficiency is usually greater than the design flowrate. As shown in Figure 5, the predictions by the empirical correlation method have large deviations with the experimental values. The ELM/CFD method significantly improves the prediction accuracy. At \( Q = 4 \text{ m}³·\text{h}^{-1} \), when the efficiency is the highest, the deviations of the pump head and the efficiency are both less than 2%, and under the design flow are both less than 4%. Moreover, within the range of 50% to 185% of the design flow, the deviations of the head and the efficiency are within 2% and 5%, respectively. The comparison indicates that the ELM/CFD method is accurate and reliable in predicting the performance of the high-speed centrifugal pumps with the low specific speed.

Table 3. Measurement results of low specific speed high-speed centrifugal pump.

<table>
<thead>
<tr>
<th>( Q/\text{m}³·\text{h}^{-1} )</th>
<th>( H/\text{m} )</th>
<th>( P/\text{W} )</th>
<th>( \eta_{\text{all}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.63</td>
<td>16.60</td>
<td>804</td>
<td>31.62%</td>
</tr>
<tr>
<td>5.37</td>
<td>18.31</td>
<td>801</td>
<td>33.38%</td>
</tr>
<tr>
<td>4.92</td>
<td>20.57</td>
<td>788</td>
<td>34.92%</td>
</tr>
<tr>
<td>4.54</td>
<td>22.47</td>
<td>774</td>
<td>35.83%</td>
</tr>
<tr>
<td>3.99</td>
<td>24.78</td>
<td>749</td>
<td>35.89%</td>
</tr>
<tr>
<td>3.10</td>
<td>27.55</td>
<td>696</td>
<td>33.40%</td>
</tr>
<tr>
<td>2.90</td>
<td>28.00</td>
<td>693</td>
<td>31.89%</td>
</tr>
<tr>
<td>2.14</td>
<td>29.72</td>
<td>645</td>
<td>26.81%</td>
</tr>
<tr>
<td>1.80</td>
<td>30.28</td>
<td>620</td>
<td>23.90%</td>
</tr>
<tr>
<td>1.42</td>
<td>30.76</td>
<td>596</td>
<td>19.94%</td>
</tr>
<tr>
<td>0.95</td>
<td>31.51</td>
<td>578</td>
<td>14.09%</td>
</tr>
</tbody>
</table>

Figure 5. Comparison between numerical simulation results and experimental data.
3.2. Flow Field Analysis of the High-Speed Centrifugal Pump

The variations of the total pressure and static pressure in the main channel along the streamline are shown in Figure 6. At Mark 1 in Figure 6, the fluid flows into the inducer from the inlet section and the pressure increases under the pre-pressurization of the inducer. With the increasing flowrate, the increasing speed decreases continuously, even a drop occurs at \( Q = 5 \text{ m}^3\text{h}^{-1} \). The fluid comes out of the inducer and flows into the impeller channel. As the blade does not cover the whole channel, the fluid first enters a leafless area, where the static pressure is bound to be reduced or even become negative and the cavitation is most likely to happen. It can be seen from Figure 6 that with the increase of flow, the negative pressure area expands and the minimum static pressure decreases. Moreover, the dynamic pressure loss in the vaneless diffuser increases. Under the small flowrate and the design flowrate, the pre-pressurization effect of the inducer is well and the static pressure value is reasonable.

After the fluid enters the area with blades, the pressure rises rapidly and it fluctuates after rising at the junction of the impeller and the vaneless diffuser, as shown in Mark 2 in Figure 6. This is because there is the reflux and hub leakage between the impeller and the vaneless diffuser. The reflux will cause the total pressure fluctuation in some areas. The hub leakage is the result of the high-pressure fluid flow, which then flows into the impeller. As a result, the total pressure rises to a certain extent. It can be seen from Figure 7 that the distribution of streamline near the impeller outlet is relatively uniform. At \( Q = 1 \text{ m}^3\text{h}^{-1} \), the flow is blocked by the blades when flowing into the vaneless diffuser and forced to divert, resulting in a serious reflux area in the impeller on the pressure surface of the blade. As the flow increases, the flow velocity vector increases while the shock caused by the blade relatively decreases. Under the rated condition, there is still a slight disturbance in the streamline on the blade’s leading edge. When the flowrate continues to increase, there is no more eddy current phenomenon. According to the above analysis, the reflux phenomenon on the interface gradually disappears and the dynamic pressure loss accordingly decreases, resulting in the alleviation of the “fluctuation” phenomenon of total pressure. The liquid flows into the vaneless diffuser after flowing out of the curved vaneless diffuser, whose hydraulic loss increases as the flow increases, as shown in Mark 3 in Figure 6. The pressure distribution of the vaneless diffuser is reasonable under the small flowrate and the design flowrate. The drastic change in the cross-sectional area at the interface between the diffuser outlet and motor flow channel leads to the loss of various reflux and secondary flow. At Mark 4 and Mark 5 in Figure 6, which indicates the inlet of the bearing channel, the total pressure fluctuates slightly as the flowrate is three orders of magnitude smaller than the mainstream, so the disturbance is particularly small.
3.3. Loss Analysis Based on ELM/CFD Method

Based on the numerical results, the energy power loss and the proportion of energy loss power are calculated by the equations in 2.4. The variations of different power losses with the flow rate are shown in Figure 8. Except for the useful power, the hydraulic loss of impeller is the largest loss in the pump, followed by the volume loss power, the hydraulic loss of motor channel, and the mechanical loss power. With the increase of flow, both the total power and the top curve first increase and then decrease after $Q = 4.2 \text{ m}^3\text{ h}^{-1}$. Meanwhile, the lowest curve representing the useful power increases and finally tends to keep constant. The disc friction loss power basically maintains constant and the change of flow almost has little effect on it. The hydraulic power loss, the hydraulic loss of the inducer is the smallest, while that of the impeller is the largest. Moreover, the hydraulic loss of the impeller decreases significantly when the flowrate is greater than $4 \text{ m}^3\text{ h}^{-1}$. The hydraulic loss in the diffuser is almost zero under smaller flow rates and it increases rapidly under large flowrates. The hydraulic loss in the motor channel increases slightly as the flow increases.

The analysis of the proportion of energy loss under the design condition and the deviation condition is shown in Figure 9. At $Q = 3 \text{ m}^3\text{ h}^{-1}$, the useful power accounts for the highest proportion, 34.2%, which means the pump efficiency is 34.2%. The largest loss is the hydraulic loss of the impeller at 183.0 W, accounting for 27.3%, followed by the volume loss at 93.1 W, accounting for 13.9%. The disc friction loss is 52.2 W and the hydraulic loss of the motor channel is 50.1 W, which are relatively small. Meanwhile, the others can be ignored for they are all below 4%. To further improve the pump efficiency, the hydraulic loss of the impeller should be reduced. Increasing the number of blades and modifying the inlet and outlet angle of the blades can improve the distribution of the impeller flow field, but it will also increase the impact loss at the inlet of the impeller.
When the pump works in the low flow rate, the hydraulic loss of the impeller is up to 177.8 W, accounting for 35.4%, and the volume loss is up to 115.4 W, accounting for 23.0%, which is much higher than the useful power at 85.4 W. Under the working condition, the pump efficiency is only 17.0%. The hydraulic loss power and its proportion of other flow channels are lower than that under the design conditions, indicating that the factors in restricting the efficiency of the centrifugal pump under small flow rates are the “blockage” phenomenon of the centrifugal impeller and the large leakage flow. To optimize the performance of the pump, the impeller structure should be improved. Increasing the blade wrap angle and the number of blades and adopting the composite impeller structure works. Moreover, reducing the clearance between the tip and the hub as much as possible is also a choice. However, the methods above will be limited by the processing conditions and there will be a risk of strong pressure pulsation.

When the pump works in a high flow rate, the useful power is 271.9 W, accounting for 36.4%, which is much higher than other losses, and the pump efficiency is 36.4%. The hydraulic loss of the impeller is 104.0 W, accounting for 13.9%, and the volume loss is 68.2 W, accounting for 9.1%, both of which are much lower than that under the design condition. However, the hydraulic loss of the other channels increases rapidly, indicating that the factor in restricting the efficiency of the pump in this study is the increment of resistance loss along other tracks caused by the excessive flow rate.

4. Conclusions

In this paper, CFD simulations and the energy loss model are combined to calculate different power losses and to analyze the performance of the high-speed centrifugal wa-
ter pump with the low specific speed. The simulation results are validated against the experimental results. The main conclusions of this paper are as follows.

1. ELM/CFD can better predict the performance of the high-speed centrifugal pump with the low specific speed. Comparing the experimental results with the calculated results, it is found that the results of ELM/CFD are more consistent with the experimental values, with the relative deviation of pump head and efficiency less than 2% and 5% under 50% to 185% design flowrate, respectively.

2. Under different flowrates, the variations of the pressure in the high-speed centrifugal pump along the streamline are the same, but the amplitude is quite different. As the flowrate increases, the low-pressure area expands and the minimum static pressure decreases. Meanwhile, the reflux phenomenon at the junction of the hydraulic components alleviates.

3. ELM/CFD can accurately analyze the optimization direction of the pump. As for the high-speed centrifugal water pump with the low specific speed, the factors in restricting the efficiency of the pump under smaller flowrates are the hydraulic loss and volume loss of the impeller, and the centrifugal pump under large flowrates could be optimized by reducing the hydraulic loss of the subsequent channel.

As for the low specific speed pump with a fully sealed structure, the empirical correlation method ignores the influence of leakage and overestimates the pump performance. The ELM method is mainly based on empirical parameters, which will affect the accuracy of the performance prediction. The ELM/CFD method can obtain more accurate prediction of the performance of the high-speed centrifugal pump than the empirical correlation method, and this method more directly reflects the optimization direction. The research results in this study could provide references on the practical engineering application of the performance prediction and the optimization of pumps.

Author Contributions: Data curation, X.L. and B.Z.; investigation, X.L.; project administration, R.X.; software, H.Z.; supervision, R.X. and Y.H.; writing—original draft, X.L. and B.Z.; writing—review and editing, T.L. All authors have read and agreed to the published version of the manuscript.

Funding: This project is supported by the National Natural Science Foundation of China (51976150) and the Youth Innovation Team of Shaanxi Universities.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: Not applicable.

Conflicts of Interest: The authors declare no conflict of interest.

References


