The Effect of Different Inflows on the Unsteady Hydrodynamic Characteristics of a Mixed Flow Pump

Long Yun¹, Wang Dezhong¹, Yin Junlian¹, Cai Youlin² and Feng Chao²

¹ School of Mechanical Engineering, Shanghai Jiao Tong University
800 Dongchuan Road, Shanghai, 200240, China, longyunjs@sjtu.edu.cn, dzwang_sjtu@sina.com
² Marine Design & Research Institute of China
NO 1688 South Xizang Road Shanghai, 200011, China, maricjet@maric.com.cn

Abstract

The problem of non-uniform inflow exists in many practical engineering applications, such as the elbow suction pipe of waterjet pump and, the channel head of steam generator which is directly connect with reactor coolant pump. Generally, pumps are identical designs and are selected based on performance under uniform inflow with the straight pipe, but actually non-uniform suction flow is induced by upstream equipment. In this paper, CFD approach was employed to analyze unsteady hydrodynamic characteristics of reactor coolant pumps with different inflows. The Reynolds-averaged Naiver-Stokes equations with the k-ε turbulence model were solved by the computational fluid dynamics software CFX to conduct the steady and unsteady numerical simulation. The numerical results of the straight pipe and channel head were validated with experimental data for the heads at different flow coefficients. In the nominal flow rate, the head of the pump with the channel head decreases by 1.19% when compared to the straight pipe. The complicated structure of channel head induces the inlet flow non-uniform. The non-uniformity of the inflow induces the difference of vorticity distribution at the outlet of the pump. The variation law of blade to blade velocity at different flow rate and the difference of blade to blade velocity with different inflow are researched. The effects of non-uniform inflow on radial forces are absolutely different from the uniform inflow. For the radial forces at the frequency $f_R$, the corresponding amplitude of channel head are higher than the straight pipe at 1.0$\Phi_d$ and 1.2$\Phi_d$ flow rates, and the corresponding amplitude of channel head are lower than the straight pipe at 0.8$\Phi_d$ flow rates.

Keywords: hydrodynamic, radial force, mixed flow pump, channel head, non-uniform.

1. Introduction

The problem of non-uniform inflow exists in many practical engineering applications, such as the elbow suction pipe of waterjet pump and, the channel head of steam generator which is directly connect with reactor coolant pump. Generally, pumps are identical designs and are selected based on performance under uniform inflow with the straight pipe, but actually non-uniform suction flow is induced by upstream equipment. The reactor coolant pump is one of the most important equipment in nuclear power plant. In the APWR reactor primary coolant system, two canned motor pumps are directly attached to the cold side of the steam generator [1], as shown in Fig. 1. The pumps are identical designs and are selected based on performance under uniform inflow with the straight pipe, but actually non-uniform suction flow is induced in the discharge pipe of the steam generator due to the complex geometry in the channel head, which might influence the performance of the pumps [2-7]. Unsteady flow problems in pumps may resonate with an acoustic mode of the inlet or discharging piping to produce a serious pulsation problem [3]. Furthermore, an effective approach should be developed to research unsteady hydrodynamic of the pumps with uniform and non-uniform inflow, which is sorely necessitated.

Generally, unsteady pressure pulsation and hydrodynamics induced by rotor–stator interaction, which excites mechanical vibrations of the pump, cannot be avoided entirely, even at the design operating condition. Up to now, lots of numerical and experimental investigations have been carried out, and the fast Fourier transform (FFT) method has been proved to be the most powerful tool to unveil the frequency characteristics of pressure fluctuations. Yao et al. [8] analysed both frequency domain and time-frequency domain in a double suction centrifugal pump based on fast Fourier transform (FFT) and time-frequency representation methods. Parrondo et al. [9] also investigated the unsteady pressure distribution in a conventional centrifugal pump, and particular attention was paid on pressure fluctuations at blade passage frequency. Some studies have laid their emphasis upon the influence of
geometrical parameters on pressure pulsation characteristics by either experimental or numerical methods. Spence et al. [10] explored the effects of pressure pulsation. In their work, they took the form of a parametric study covering four geometric parameters by numerical analysis. And a rationalisation process aimed at reducing vibration through reductions in pressure pulsations has produced geometric recommendations. Yang et al. [11, 12] investigated unsteady pressure fields of the pump as turbine by numerical methods and illustrated that increasing blade tip clearance serves as an effective measure for reducing pressure pulsation. Benra et al. [13] investigated the periodic unsteady flow in a single-blade pump by CFD simulation and particle image velocimetry measurement method, and the results showed that transient numerical simulations compare very well to velocity measurements. Pei et al. [14, 15] conducted the numerical investigation on the periodically unsteady flow of a single-blade pump and predicted the flow in a whole passage. Toussaint et al. [16] conducted experimental investigation on the unsteady flow in a centrifugal pump in off-design operating conditions and concluded that the pressure fluctuations took place at blade passing frequency, rotation frequency, and their harmonics. Zhang et al. [17] explored an effective method to reduce high pressure pulsation level by investigating a slope volute pump, and analysed its influence on flow structures using numerical simulation.

Therefore, the major aim of this study is to employed CFD approach to simulate the unsteady flow field of a scaled model reactor coolant pump with uniform and non-uniform inflow conditions, then to investigate the unsteady radial forces of impeller. The three-dimensional pump internal flow channel was modelled by pro/E software, Reynolds-averaged Naiver-Stokes equations with the k-ε turbulence model were solved by the computational fluid dynamics software CFX to conduct the steady and unsteady numerical simulation, by which the flow field and radial force were obtained. Detailed analysis of radial force spectrum is performed by Fast Fourier Transform (FFT) method. The special attention is paid to compare radial force characteristics value with different inflows. Part of the work, results, methods and concepts presented in the paper have been presented and discussed at the 28th IAHR Symposium on Hydraulic Machinery and Systems, Montreal, 2014, website [http://www.iahr-grenoble2016.org/](http://www.iahr-grenoble2016.org/).

2. Numerical Simulation

2.1 The pump model

The two model pumps are identical designs, and the parameters of the pump are shown in Table 1. Nominal flow rate and head of the model pump is 950 m³/h and 13.5 m, respectively. Fig. 1 shows the cold side of the steam generator with two discharge pipes. Since the cold side of the steam generator is symmetrical, it is assumed that the flow field inside is also symmetrical. So the channel head could be divided into two mirror parts. The effect of the channel head and straight pipe on the performance of the pump can be investigated individually. The computational domain includes the channel head, straight pipe, impeller, diffuser, spherical casing with straight outlet pipe, as shown in Fig. 2(a) and Fig. 2(b). Both the X radial force and the Y radial force of the impeller are monitored.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impeller outlet diameter $D_2$</td>
<td>271.5 mm</td>
<td>Tangential velocity of impeller outlet $u_2$</td>
<td>21.03 m/s</td>
</tr>
<tr>
<td>Impeller outlet width $b_2$</td>
<td>83 mm</td>
<td>Impeller blade number $Z_1$</td>
<td>5</td>
</tr>
<tr>
<td>Rotating speed $n_d$</td>
<td>1480 r/min</td>
<td>Diffuser blade number $Z_2$</td>
<td>11</td>
</tr>
<tr>
<td>Angular velocity $\omega$</td>
<td>155 rad/s</td>
<td>Nominal flow rate coefficient $\Phi_d$</td>
<td>$Q_d/(\omega D_2^2 b_2)$</td>
</tr>
<tr>
<td>Inlet pipe diameter $D_1$</td>
<td>300 mm</td>
<td>Nominal head coefficient $\Psi_d$</td>
<td>$gH_d/(\omega^2 d_2^2)$</td>
</tr>
<tr>
<td>Pressure amplitude $p$</td>
<td>Pa</td>
<td>Specific speed</td>
<td>387.5</td>
</tr>
<tr>
<td>Pressure coefficient $C_p$</td>
<td>$p/0.5 \rho u_c^2$</td>
<td>Water density $\rho$</td>
<td>1000 kg/m³</td>
</tr>
<tr>
<td>Rotating frequency of impeller $f_R$</td>
<td>24.7 Hz</td>
<td>Stator passing frequency $f_{SPF}$</td>
<td>271.3 Hz</td>
</tr>
</tbody>
</table>

2.2 The mesh

In order to improve the grids quality, nearly all the computational domains are meshed by the hexahedral elements, except for the channel head with complex geometry. The grids of the impeller, diffuser and casing are shown in Fig. 3. The magnitude of y⁺ around the blades is lower than 200. The straight pipe grid number is 1,214,400, the channel head grid number is 1,967,476, impeller grid number is 1,132,405, the diffuser grid number is 1,919,775, and the casing grid number is 1,276,405.
The computational domains and the monitor points

The head curve of the pump

Grids of each component

2.3 The boundary condition

To obtain a stable numerical simulation, an initial value distribution of the flow parameters used as exactly as possible was required. The working medium is water at 25°C. The rotation of the impeller is set to be 1480r/min. The mass flow rate at inlet and the pressure at outlet are set as the boundary condition. The k-ε turbulent model is chosen to solve RANS equation. The adiabatic and no-slip boundary condition is applied to the solid walls. The residual convergence precision of the steady calculation is set to 10⁻⁴. The second-order backward Euler scheme is chosen for the time discretization. The interface between the impeller and the casing is set to “transient rotor-stator” to capture the transient rotor-stator interaction in the flow. The chosen time step ∆t for the transient simulation is 3.3784×10⁻⁴ s for the nominal rotating speed, corresponding to the changed angle of 3° of impeller rotation. Therefore, 120 transient results are included for one impeller revolution calculation. Ten revolutions of the impeller for the design condition are conducted. Within each time step, the number of iterations has been chosen to 25 and the iteration stops when the maximum residual is less than 10⁻⁴.

3. Results and Discussions

3.1 Hydraulic performance

Figure 2(c) shows the pump hydraulic performance of different suction flows and experiment, and the experiment is conducted with the straight pipe. The numerical results of the straight pipe and channel head were validated with experimental data for the heads at different flow coefficients. The heads of the pump at 0.8\(\Phi_d\), 1.0\(\Phi_d\), 1.2\(\Phi_d\) compared with CFD and experiment were shown in Table 2. \(H_C\) means the head of the pump with the channel head, \(H_S\) means the head of the pump with the straight pipe, \(H_{exp}\) means the head of the pump by experiment. It can be concluded that the suction flow coming from the channel head of the steam generator has a considerable effect on the performance of the canned motor pumps.

<table>
<thead>
<tr>
<th>(\Phi_d/\Phi_d)</th>
<th>(H_C(m))</th>
<th>(H_S(m))</th>
<th>(H_{exp}(m))</th>
<th>(H_C-H_{exp})/(H_{exp})(%)</th>
<th>(H_S-H_{exp})/(H_{exp})(%)</th>
<th>(H_C-H_S)/(H_S)(%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.8</td>
<td>17.365</td>
<td>16.878</td>
<td>17.195</td>
<td>-2.804</td>
<td>-0.978</td>
<td>-1.844</td>
</tr>
<tr>
<td>1.0</td>
<td>15.400</td>
<td>14.859</td>
<td>15.039</td>
<td>-3.511</td>
<td>-2.347</td>
<td>-1.192</td>
</tr>
<tr>
<td>1.2</td>
<td>11.801</td>
<td>11.629</td>
<td>12.031</td>
<td>-1.460</td>
<td>1.952</td>
<td>-3.347</td>
</tr>
</tbody>
</table>

3.2 Flow field analysis

To clarify the reason leading to the difference of pump performance with different inflow, Fig. 4 presents the velocity streamline in the channel head. In figure 5, at nominal flow rate, due to the complicated structure of channel head, the velocity streamline turn suddenly at the throat, which make the pump inlet exists the low velocity region, as shown in figure 6. Therefore the larger vorticity occurs in this low velocity region, as shown in figure 7. Because of the non-uniform inflow, the vorticity distribution at the outlet of the pump connected with channel head is different from the straight pipe. Besides, the velocities of blade to blade in the impeller with different suction flows from 0.8\(\Phi_d\) to 1.2\(\Phi_d\) conditions are shown in figure 9. Form figure 9(a), when the pump is connected with channel head, the velocity of blade to blade are non-uniform, but the level of non-uniformity decreases with the flow rate decreasing. That means the effect of non-uniform inflow on the velocity of blade to blade weakens when the flow rate decreases. Form figure 9(b),
when the pump is connected with straight pipe, the velocity of blade to blade are uniform relatively. The reason for the variation of pump performance is probably related to inflows structures of the pump with the channel head and straight pipe.

![Velocity streamline in the channel head](image1)

**Fig. 5** The velocity streamline in the channel head

![Velocity distributions at the pump inlet with the channel head](image2)

**Fig. 6** The velocity distributions at the pump inlet with the channel head

![Vorticity distributions at the pump inlet with the channel head](image3)

**Fig. 7** The vorticity distributions at the pump inlet with the channel head

![Vorticity distribution at the outlet with different inflow](image4)

**Fig. 8** The vorticity distribution at the outlet with different inflow
3.3 Unsteady pressure pulsation at the inlet and outlet of the model pump

Unsteady hydrodynamic simulations of the model pump were conducted at 0.8$\Phi_d$, 1.0$\Phi_d$, and 1.2$\Phi_d$ flow rates. To evaluate radial force energy in particular frequency band, detailed analysis of radial force spectrum is performed by Fast Fourier Transform (FFT) method. Figure 10(a) shows the X radial force and the Y radial force of the impeller with different suction flows at 0.8$\Phi_d$ flow rate. From figure 10(a), when the pump is connected with channel head, the non-uniform inflow leads to X radial force shifting to the upside between 270° and 90°, and the non-uniform inflow leads to the Y radial force shifting to the left side between 180° and 360°. Figure 10(b) shows the X radial force and the Y radial force of the impeller with different suction flows at 1.0$\Phi_d$ flow rate. From figure 10(b), when the pump is connected with channel head, the non-uniform inflow leads to X radial force shifting to the upside between 180° and 360°, and the non-uniform inflow leads to the Y radial force shifting to the left side between 90° and 270°. Figure 10(c) shows the X radial force and the Y radial force of the impeller with different suction flows at 1.0$\Phi_d$ flow rate. From figure 10(c), when the pump is connected with channel head, the non-uniform inflow leads to X radial force shifting to the upside between 180° and 360°, and the non-uniform inflow leads to the Y radial force shifting to the left side between 60° and 240°. And this offset leads to the asymmetry of radial force increase. It will increase the risk of rotor-bearing system.
Meanwhile, the non-uniform inflow also leads to the differences of radial force spectrum. Figure 11 shows the radial force frequency domain of the impeller with different suction flows at 0.8\( \Phi_d \), 1.0\( \Phi_d \) and 1.2\( \Phi_d \) flow rates. From figure 11, the predominant components in force spectra locate at \( f_{SP} \), where the corresponding amplitudes of straight pipe and channel head are almost same, and the secondary components in force spectra locate at \( f_R \). From figure 11(a), at the frequency \( f_R \), the corresponding X radial force amplitude of channel head is 63.90N, and the corresponding X radial force amplitude of straight pipe is 106.13N. So the corresponding amplitude of channel head decrease by 39.79\%. The corresponding Y radial force amplitude of channel head is 65.285N, and the corresponding Y radial force amplitude of straight pipe is 119.56N. So the corresponding amplitude of channel head decrease by 45.40\%. From figure 11(b), at the frequency \( f_R \), the corresponding X radial force amplitude of channel head is 142.46N, and the corresponding X radial force amplitude of straight pipe is 65.08N. So the corresponding amplitude of channel head increase by 118.90\%. The corresponding Y radial force amplitude of channel head is 142.14N, and the corresponding Y radial force amplitude of straight pipe is 64.61N. So the corresponding amplitude of channel head increase by 120\%. From figure 11(c), at the frequency \( f_R \), the corresponding X radial force amplitude of channel head is 152.15N, and the corresponding X radial force amplitude of straight pipe is 59.46N. So the corresponding amplitude of channel head increase by 155.89\%. The corresponding Y radial force amplitude of channel head is 153.10N, and the corresponding Y radial force amplitude of straight pipe is 60.02N. So the corresponding amplitude of channel head increase by 155.08\%. In a word, at the frequency \( f_R \), the corresponding amplitude of channel head are higher than the straight pipe at 1.0\( \Phi_d \) and 1.2\( \Phi_d \) flow rates, and the corresponding amplitude of channel head are lower than the straight pipe at 0.8\( \Phi_d \) flow rates. That means the non-uniform inflow increases the radial force at low frequency in larger flow rate condition, which may leads to the low-frequency vibration of the shaft system.
4. Conclusion

In this paper, the numerical analysis on unsteady hydrodynamic features of a scaled model reactor coolant pump with different inflows is presented. The pump head with the straight pipe measured by experimental method is compared to that of the channel head and the straight pipe obtained by numerical method. The analysis focuses on the flow fields and radial force of the impeller in different flow rate conditions. Detailed analysis of radial force spectrum is performed by Fast Fourier Transform (FFT) method.

The numerical results of the straight pipe and channel head were validated with experimental data for the heads at different flow coefficients. In the nominal flow rate, the head of the pump with the channel head decreases by 1.19% when compared to the straight pipe. The complicated structure of channel head induces the inlet flow non-uniform, such as the velocity streamline, vorticity, velocity. The non-uniformity of the inflow induces the difference of vorticity distribution at the outlet of the pump connected with the channel head and straight pipe. The variation law of blade to blade velocity at different flow rate and the difference of blade to blade velocity with different inflow are shown.

The effects of non-uniform inflow on radial forces are absolutely different from the uniform inflow. For the radial forces at the frequency $f_R$, the corresponding amplitude of channel head are higher than the straight pipe at 1.0$\Phi_d$ and 1.2$\Phi_d$ flow rates, and the corresponding amplitude of channel head are lower than the straight pipe at 0.8$\Phi_d$ flow rates.

Acknowledgments

This work is funded by Shanghai Economy Information Technology Committee and National Natural Science Foundation of China (No.51576125).

Reference