Blade rows interaction in contra-rotating axial flow pump designed with different rotational speed concept

L L Cao1, S Watanabe2, T Imanishi1, H Yoshimura1 and A Furukawa3

1 Graduate School of Engineering, Kyushu University, Kasuga, Fukuoka 816, Japan
2 Department of Mechanical Engineering, Kyushu University, Kasuga, Fukuoka 816, Japan
3 Oita National College of Technology, 1666 Maki, Oita, 870-01 Japan

E-mail: cao.linlin.711@s.kyushu-u.ac.jp

Abstract. The contra-rotating axial flow pump consisting of counter-rotating tandem rotors has been experimentally confirmed with better performances than the conventional axial flow pump, but it is known to suffer from the significant potential interaction between the counter-rotating blade rows, which is responsible for the repetitive stresses and unfavourable to the reliable operation. Consequently, to improve the reliability of contra-rotating axial flow pump including the reduction of the blade rows interaction, a new type of rear rotor was designed in the previous study by the rotational speed optimization methodology with some additional considerations. In the present study, to understand the effectiveness of the new design method, instantaneous static pressure fluctuations on the casing wall under the design and off design conditions are investigated by means of experimental and numerical simulation methods. The Fourier analysis is employed to process the data obtained from experiments and numerical simulations, and the axial distribution of the Blade Passing Frequency (BPF) amplitude is obtained. The new rear rotor shows weakened BPF amplitude both upstream and downstream especially at the positions between the two blade rows in both CFD and EFD analyses, implying the reduced blade rows interaction with the new rear rotor.

1. Introduction

The axial flow pump with higher flow rate operation is increasingly required in several industrial fields due to its advantages of the low cost and compact structure [1]. The contra-rotating axial flow pump with two rotors rotating reversely, offers a better solution for these fields [2]. Actually, in our previous experiments, it has been confirmed that, the contra-rotating axial flow pump has better hydraulic and cavitation performances than the conventional axial flow pump with the same design specification. At the off design range, its efficiency can be widely improved by separated rotational speed control on each rotor [3,4].

In one of our prototype contra-rotating rotors, where the rear rotor, herein named RR2, was designed with the same design rotational speed as the front rotor \((N_f = N_r)\) by the conventional design method of axial flow rotor, it was experimentally observed that the measured performances failed to satisfy the design specifications due to the complexity of the flow between the front and rear rotors. In addition, the cavitation inception was observed in the tip region of RR2 rather than the front rotor at the off design conditions, indicating large static pressure reduction in the mid positions of the rotor pair [3]; the significant blade rows interaction was observed, which was responsible for the repetitive stresses and unfavourable to the safe and reliable operation [5]. Consequently, in our previous paper,
the rotational speed optimization methodology was proposed with some additional considerations, aiming at reduction of blade rows interaction, better cavitation performance and the secondary flow suppression [6]. Given the optimized velocity triangles and the identical front rotor, a new rear rotor RR3 was designed with the reduced rotational speed \( N_r < N_f \). It was confirmed both experimentally and numerically that the newly designed rotor pair (RR3 pair) highly satisfies the design specification, indicating the validity of the rotational speed optimization methodology in terms of performances at the design condition [6].

In the present study, for the further understanding of the effectiveness of the new design method, instantaneous static pressure fluctuations on the casing wall under the design and off design conditions are investigated by means of experimental and CFD approaches. In the experiment, the Fourier analysis is employed in processing the measured static pressure data, which are sampled in time domain at several axial locations on the casing wall. In the CFD study, the unsteady simulation is carried out, but to save the computation resources, the Fourier’s analysis is carried out on the pressure information circumferentially sampled in space domain at several axial locations. Both the conventional and newly designed rotors RR2 and RR3 are analyzed by these methods to investigate the effectiveness of rotational speed optimization method in terms of the reduction of blade row interaction.

2. Geometrical and performance characteristics of test rotors RR2 and RR3

Compared to the prototype rotors designed with the same rotational speed as \( N_f = N_r = 1225 \) rpm, the rotational speed combination of the new rotor pair is determined as \( N_f = 1311 \) rpm and \( N_r = 1123 \) rpm by the following equation (1) & (2), in which the equivalent inflow relative velocity \( W_f = W_r = W_{i_f} \) is assumed for both front and rear rotor with satisfying the design specifications of the total head of \( H = 4 \) m and the flow rate of \( Q = 70 \) L/s.

\[
N_f = \frac{60}{\pi D_f} \sqrt{\frac{gH/\eta}{1 - \left(1 - \psi_f/\eta_f\right)^2}}
\]

\[
N_r = \frac{60}{\pi D_f} \left(1 - \frac{\psi_f}{\eta_f}\right) \sqrt{\frac{gH/\eta}{1 - \left(1 - \psi_f/\eta_f\right)^2}}
\]

The brief parameters of five spanwise sections of both rear rotors’ blades are listed in table 1. The both rear rotors have 5 blades against 4 blades of the front rotor. The RR3 with the rotational speed optimization method applied shows lower stagger angle \( \gamma \) at most of span compared with the conventional type RR2, partly aiming at reducing the potential interaction from the rear rotor to the front rotor.

| Blade Profile | Section | 1(Hub) | 2 | 3 | 4 | 5(Tip) |
|---------------|---------|--------|---|---|---|--------|
| RR2 | \( x/l \) | 0.40 | 0.40 | 0.40 | 0.40 | 0.40 |
| | \( \gamma [^\circ] \) | 64.24 | 68.86 | 72.54 | 75.34 | 77.56 |
| | \( \alpha [^\circ] \) | 5.17 | 3.08 | 1.56 | 0.53 | -0.23 |
| RR3 | \( x/l \) | 0.60 | 0.55 | 0.50 | 0.45 | 0.40 |
| | \( \gamma [^\circ] \) | 64.95 | 67.05 | 69.73 | 72.33 | 71.88 |
| | \( \alpha [^\circ] \) | 3.414 | 3.793 | 3.292 | 2.529 | 8.613 |

The measured performances of the separate rotor and the whole pump are plotted for the prototype pair RR2 and the new pair RR3 in figure 1, where the head and flow coefficients are defined respectively by \( \psi = gh/Ut^2 \) and \( \phi = Q/(\pi r_c^2 - r_h^2) Ut \), and the subscripts ‘new’ and ‘D’ denote new designed rotor pair and design condition respectively. It is obvious that with the new rear rotor RR3 applied, the
whole pump and each rotor show favorable performances in both experimental and CFD analyses at the design flow rate compared with the RR2 pair, and the improvements in the normalized performances of the identical front rotor imply the weakened potential interaction between the new rotor pair.

![Performance Graphs](image)

**Figure 1.** Head and efficiency performances of rotors for both RR2 and RR3 pairs

Despite of the superior performances at the design condition, a more pronounced positive slope appears on the new rear rotor’s Head-Flow rate curve at the partial flow rate \( Q \leq 28 \text{L/s} \), resulting in the inferior head and efficiency performance of the whole pump in this flow rate region. Combined with the previous experimental and CFD results, the flow rate \( Q=28 \text{L/s} \) is considered as the critical point where the significant variation of the internal flow structure occurs. Therefore, in the present study, the casing wall static pressure distribution at this critical flow rate in addition to the design flow rate is investigated to offer more evidences for the blade rows interaction and the mechanism of the positive slope on the performance curves.

3. **Experimental and numerical investigation methods for pressure fluctuations**

3.1. Static wall pressure sampling method in experiments

Since we have already carried out the pressure measurement of the RR2 rotor in our previous paper [5], we herein concentrate on the measurement of the pressure fluctuations in the RR3 rotor pair.

The pressure fluctuations on the casing wall are measured at eight axial locations separately for the front and rear rotors as shown in figure 2. The sampling frequency is determined as every 2.5 degree rotation of the front rotor which rotates faster than the rear rotor in the RR3 pair, resulting in the sampling time interval as 3.18e-4 second with \( N_f =1311 \text{ rpm} \). The total sampling number is determined as 8192, which provides us with the sufficient data in time domain to carry out the Fourier analysis.

![Casing Wall Locations](image)

**Figure 2.** Axial locations of two groups of eight pressure taps for the measurement of the casing wall instantaneous static pressure, which covers the upstream, leading edge, trailing edge and downstream of each rotor.
3.2. Static wall pressure sampling method in CFD simulation
The unsteady simulation using a commercial CFD software ANSYS CFX 13 is carried out to analyze the pressure fluctuations in both RR2 and RR3 rotor pairs. The basic settings are described in our previous papers [6]. Since it is unrealistic to obtain sufficient static pressure information in time domain by the unsteady CFD simulation due to the computation resource limitation, the static pressure information is circumferentially sampled in space domain at several certain axial positions. The calculation results at certain representative timesteps have been chosen, and then circumferentially 360 static pressure data are obtained for each cross section.

Circumferential pressure distribution is now converted to discrete Fourier series with the rotational periodicity of 360 degrees as follows:

\[ f(\theta) = a_0/2 + \sum_{k=1}^{\infty} (a_k \cos k\theta + b_k \sin k\theta) \]  

\[ A_k = \sqrt{(a_k^2 + b_k^2)} \]

Where \(a_k = \frac{1}{\pi} \int_{-\pi}^{\pi} f(\theta) \cos k\theta \, d\theta\) and \(b_k = \frac{1}{\pi} \int_{-\pi}^{\pi} f(\theta) \sin k\theta \, d\theta\)

Since the blade number of the front rotor and rear rotor are 4 and 5 respectively, the amplitudes of BPF for each blade are determined by \(A_4\) and \(A_5\).

4. Results and discussion

4.1. Axial distributions of averaged static pressures

Figure 3 represents the axial distributions of the circumferentially averaged static head rise \(H_s\) from the first measurement position “channel 1” as shown in figure 1 for the newly designed rotor pair RR3 and the conventional RR2 type, where “CFD” and “EFD” denote the results from numerical simulation and experiments data respectively. Despite of the quantitative deviation between the simulation and experimental results, the both investigation methods show the same variation tendency of the static pressure on the casing wall in the axial direction. At the design condition of \(Q=70\) L/s, the casing wall static pressure is nearly kept constant at the downstream of the front rotor and the slight reduction is observed at the leading edge of the rear rotor, implying the weak influence from the rear rotor. On the other hand, at the partial flow rate of \(Q=28\) L/s, the casing wall static pressure shows obvious decrease at the downstream of the front rotor, especially remarkable around the leading edge of the rear rotor, indicating the severe influence on the pressure field of the front rotor from the rear rotor.

![Figure 3](image)

(a) RR2 type
(b) RR3 type

**Figure 3.** Axial distributions of time averaged static pressure for both rotor pairs

Despite of the remarkable decrease of the static pressure occurs at the mid positions of the two blade rows for the both rotor pairs at the partial flow rate, the RR3 rear rotor shows a much relieved reduction compared with the prototype RR2 pair. As shown in figure 3(a), the casing wall static pressure starts to fall around the trailing edge of the front rotor with a more pronounced reduction
slope in RR2 pair than that in RR3 pair as illustrated in figure 3(b), which implies the availability of the new design method in terms of the potential interaction reduction at the partial flow rate.

4.2. BPF analysis for both rotor pairs
Since the BPF component due to front/rear rotor appearing in the rear/front rotor’s pressure field could be regarded as the results of blade rows interactions, we herein investigate the effectiveness of the new rear rotor design methodology in terms of the reduction of blade rows interaction from the results of the Fourier analyses.

Figure 4 shows the distributions of pressure fluctuation amplitudes due to the rotating blade rows; the fluctuation with \( n=4 \) (\( f=4N_f/60 \)) corresponds to the BPF component of the front rotor while \( n=5 \) (\( f=5N_r/60 \)) corresponds to that of the rear rotor. The dashed lines denote the BPF distribution calculated from the numerical simulation results with circumferential sampling in space domain as described in Section 3.2. For both rotor pairs, the estimations based on CFD results generally reproduce the BPF distribution obtained by the experimental method. At the design flow rate of \( Q=70 \) L/s, a good agreement is found between two groups of BPF results both in quantity and tendency. At the partial flow rate of \( Q=28 \) L/s, despite of the larger quantitative deviation in some region, which is possibly related to the overestimation of internal recirculation by CFD simulation [7], the similar BPF amplitude distribution still could be recognized.

![Figure 4. Axial distributions of BPF amplitudes of pressure fluctuations for both rotor pairs](image)

For the both rotor pairs, the BPF component due to the rear rotor (\( n=5 \) component) is observed to decay much rapidly in the downstream direction and nearly vanished at the trailing edge, while it shows a much gentle decrease slope in the upstream direction as shown in figure 4 (b) and (d). This implies that the front rotor’s pressure field is strongly influenced by that of the rear rotor. From the BPF component due to the front rotor rotation (\( n=4 \) component) shown in figure 4(a) and (c), the pressure fluctuations downstream of the front rotor decay rapidly up to the same amplitude level in both RR2 and RR3 pairs, despite that the front rotor with the RR3 rear rotor has larger blade loading mainly due to the increased rotor speed than that with the RR2 rear rotor. Then, the significance of the blade rows interaction can be characterised by the BPF component due to the rear rotor rotation, arising in the front rotor field and the mid positions between the front and rear rotors.
By comparing BPF component with \( n=5 \) plotted in figure 4(b) and (d), we can notice that the new rear rotor RR3 shows much lower BPF amplitude peak at both design and off design conditions, and it is distinctive that the BPF amplitude of the new rotor decays to the much lower level at its leading edge, then continues decaying to a slightly lower level at the front rotor’s trailing edge compared with that of the prototype rear rotor RR2. This indicates much weakened influence to the pressure field of the front rotor from the new rear rotor RR3, implying the effectiveness of the reduced rear rotor speed design in terms of blade row interaction.

5. Summary
In the present study, to further understand the effectiveness of the reduced rear rotor speed design methodology on the reduction of blade rows interaction, instantaneous static pressure fluctuations on the casing wall under the design and off design conditions are investigated by means of experimental and CFD methods.

Our conclusions are briefly summarized as follows:
1) The BPF distribution obtained from the CFD results with circumferential sampling in space domain shows good agreement with that calculated from experimental data sampled in time domain. This fact offers the present estimation method of the blade rows interaction based on CFD as an available method.
2) Despite of the remarkable decrease of the static pressure from the front rotor trailing edge toward the leading edge of the rear rotor at the partial flow rate, much relieved reductions are observed for the new rear rotor with the reduced speed design compared with the prototype rotors.
3) The new rear rotor shows much lower BPF amplitude distributed in the whole axial region, especially at the mid positions of the two blade rows, indicating much weakened influence to the pressure field of the front rotor as expected in the design consideration.

Acknowledgement
The authors would like to thank to Mr. Kusuo Okuma for his sincere continuous support through this study. They also thank to Mr. Hironori Honda for the implement of the experiment.

References
[1] Wada A and Uchida S 1999 Torishima Review (in Japanese) 13 32-5.
[2] Furukawa A, Cao Y, Okuma K and Watanabe S 2000 Proc. 2nd Int. Symp. on Fluid Machinery and Fluid Engineering (Beijing, China, October 2000) 245-52.
[3] Furukawa A, Shigemitsu T and Watanabe S 2007 Journal of Thermal Science 16 7-13.
[4] Momosaki S, Usami S, Okuma K, Watanabe S and Furukawa A 2010 Proc. of 3rd Asian Joint Workshop on Thermophysics and Fluid Science (Matsue, Japan, 10-14 September 2010) No.JP-02.
[5] Shigemitsu T, Takano T, Furukawa A, Okuma K and Watanabe S 2005 Journal of Thermal Science 14 142-9.
[6] Cao L, Watanabe S, Momosaki S, Imanishi T and Furukawa A 2012 Proc. 5th Int. Symp. on Fluid Machinery and Fluid Engineering (Jeju, Korea, 24-27 October 2012) REF-1150.
[7] Cao L, Watanabe S, Imanishi T, Yoshimura H and Furukawa A 2013 Journal of Thermal Science 22(4) 345-51