Performance prediction model of contra-rotating axial flow pump with separate rotational speed of front and rear rotors and its application for energy saving operation

De ZHANG*, Yusuke KATAYAMA**, Satoshi WATANABE**, Shin-Ichi TSUDA** and Akinori FURUKAWA**

*Graduate School of Engineering, Kyushu University
744 Motooka, Nishi-ku, Fukuoka 819-0395, Japan
E-mail: fnnabe@mech.kyushu-u.ac.jp
**Department of Mechanical Engineering, Kyushu University
744 Motooka, Nishi-ku, Fukuoka 819-0395, Japan

Received: 28 February 2020; Revised: 22 June 2020; Accepted: 30 June 2020

Abstract

Compared with conventional high-specific-speed axial flow pump, better cavitation performance and compact size have been achieved in contra-rotating axial flow pump, where the rear rotor is employed additionally to the front rotor to convert the swirling flow to the pressure rise. Meanwhile, significantly deteriorated performance has also been observed at well off-design flow rates with design rotational speed. The rotational speed control (RSC) of front and rear rotors has been experimentally proved to be effective to enhance the performance. However, thorough investigations are necessary to find the optimum rotational speeds of rotors. It may be done by computational fluid dynamics (CFD) simulations, whereas it is time-consuming to cover the wide ranges of rotational speeds. Therefore, in the present paper, a fast and effective performance prediction model is established by considering radial equilibrium condition, conservation of rothalpy and mass, empirical deviation angle, blade-rows interaction and empirical losses. Experimental and CFD results are employed to validate the proposed prediction model. It is found that the proposed model shows good enough accuracy in predicting performances of contra-rotating axial flow pump under RSC in broad flow rate range. Furthermore, an energy saving application of the proposed model is also illustrated for two typical system resistances. Compared with the traditional valve control under the design rotational speed operation, the RSC method can well modify the pump head to satisfy the system resistance at wide flow rate range with the significantly improved energy performance.

Keywords: Contra-rotating axial flow pump, Rotating speed control, Performance prediction model, Computational fluid dynamics (CFD), Energy saving

1. Introduction

A demand on high-specific-speed axial flow pump has increased in the recent years (Wada and Uchida, 1999). However, conventional axial flow pump with high specific speed usually suffers from cavitation which causes many unfavorable problems such as performance deterioration, erosion, noise and vibration (Shi et al., 2017; Stepanoff, 1957). One possible solution is to employ counter-rotating rotors, which have been applied in various types of turbopumps (Kanemoto et al., 2011; Tosin et al., 2016). Recently, Furukawa et al. (2007) have achieved better cavitation performance and compact size by applying contra-rotating front and rear rotors in the high-specific-speed axial flow pump. Meanwhile, the significantly deteriorated performance has also been observed at the off-design flow rates under the design rotational speed. This implies that large amount of energy is wasted at the off-design conditions.

To enhance the performance, rotational speed control (RSC) has been applied for the front and rear rotors of contra-rotating axial flow pump in experiments and has been proven to be effective to achieve better efficiency at various flow rates (Momosaki et al., 2010a). However, the thorough investigations are necessary to find the optimum rotational speeds
at each flow rate. It may be done by the computational fluid dynamics (CFD) simulations which are known to evaluate the performance of contra-rotating axial flow pump in sufficient accuracy (Momosaki et al., 2010b), whereas it is time-consuming to cover the wide ranges of rotational speeds. Furthermore, when RSC is applied, the shaft frequency and many related components in pressure fluctuations appear due to the difference of rotational speeds of front and rear rotors (Cao et al., 2014), which may possibly coincide with the natural frequencies of pump structures. As a whole, it is really useful if one can establish a fast and effective performance prediction model to determine the optimum RSC.

The radial equilibrium condition is well employed at the inlet and outlet of rotor in the design of axial flow turbomachines (Brennen, 1994). It is also utilized that the rothalpy is ideally kept constant along a steamtube in adiabatic steady flows (Inoue and Kamata, 1989). As a result, when assuming steady and non-viscous flow and applying conservation equations of rothalpy and mass with considering empirical deviation angle equation (Lieblein, 1965), it is often possible to calculate the flow velocities and theoretical head. Furthermore, the empirical cascade loss equation (Lieblein, 1959) is also helpful in the performance evaluation of axial flow turbomachines. Besides the cascade loss, the loss due to tip clearance effect is also very significant (Lakshminarayana, 1970), which can be modelled on the basis of blade tip lift coefficient. In principle, the above calculations are applicable for contra-rotating axial flow pumps and seem to be useful to determine the pump performance in very simple way.

The main objective of this paper is to establish and verify the performance prediction model for contra-rotating axial flow pump to determine the optimum rotational speeds of rotors under RSC. In the present study, the construction of the performance prediction model is firstly demonstrated. CFD simulations are conducted to establish and validate the performance prediction model. Experimental results are also employed to validate the proposed model as well as the referred CFD simulations. Finally, an energy saving application of the proposed model is illustrated for two typical system resistances imitating some applications.

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>( C_l )</td>
<td>Lift coefficient [-]</td>
</tr>
<tr>
<td>( C_d )</td>
<td>Drag coefficient [-]</td>
</tr>
<tr>
<td>( D )</td>
<td>Casing diameter [m]</td>
</tr>
<tr>
<td>( D_{eq} )</td>
<td>Equivalent diffusion factor [-]</td>
</tr>
<tr>
<td>( g )</td>
<td>Gravity [m/s²]</td>
</tr>
<tr>
<td>( H )</td>
<td>Head [m]</td>
</tr>
<tr>
<td>( H_0 )</td>
<td>Necessary head [m]</td>
</tr>
<tr>
<td>( H_{loss} )</td>
<td>Mass-average loss head [-]</td>
</tr>
<tr>
<td>( H_R )</td>
<td>System resistance head [m]</td>
</tr>
<tr>
<td>( H_{td} )</td>
<td>Design total head [m]</td>
</tr>
<tr>
<td>( H_{th} )</td>
<td>Theoretical head [m]</td>
</tr>
<tr>
<td>( i )</td>
<td>Incidence angle [°]</td>
</tr>
<tr>
<td>( k )</td>
<td>Empirical coefficient for deviation angle [-]</td>
</tr>
<tr>
<td>( L )</td>
<td>Shaft power [W]</td>
</tr>
<tr>
<td>( N )</td>
<td>Rotational speed of rotors [min⁻¹]</td>
</tr>
<tr>
<td>( N_d )</td>
<td>Design rotational speed [min⁻¹]</td>
</tr>
<tr>
<td>( N_s )</td>
<td>Specific speed [min⁻¹, m³/min, m]</td>
</tr>
<tr>
<td>( p )</td>
<td>Static pressure [Pa]</td>
</tr>
<tr>
<td>( p_t )</td>
<td>Total pressure [Pa]</td>
</tr>
<tr>
<td>( Q )</td>
<td>Volumetric flow rate [m³/s]</td>
</tr>
<tr>
<td>( Q_d )</td>
<td>Design flow rate [L/s]</td>
</tr>
<tr>
<td>( r )</td>
<td>Local radius [m]</td>
</tr>
<tr>
<td>( T )</td>
<td>Torque of rotors [N · m]</td>
</tr>
<tr>
<td>( v_a )</td>
<td>Axial component of velocity [m/s]</td>
</tr>
<tr>
<td>( v_o )</td>
<td>Swirling component of absolute velocity [m/s]</td>
</tr>
<tr>
<td>( w )</td>
<td>Relative velocity [m/s]</td>
</tr>
<tr>
<td>( y )</td>
<td>Normalized discrepancy between theoretical heads evaluated by CFD and the model [-]</td>
</tr>
<tr>
<td>( y^+ )</td>
<td>Nondimensional distance [-]</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \beta )</td>
<td>Flow angle [°]</td>
</tr>
<tr>
<td>( \beta_b )</td>
<td>Blade angle [°]</td>
</tr>
<tr>
<td>( \gamma )</td>
<td>Blade stagger angle [°]</td>
</tr>
<tr>
<td>( \delta )</td>
<td>Deviation angle [°]</td>
</tr>
<tr>
<td>( \delta_{m2/l} )</td>
<td>Momentum thickness coefficient [-]</td>
</tr>
<tr>
<td>( \zeta_c )</td>
<td>Cascade loss coefficient [-]</td>
</tr>
<tr>
<td>( \zeta_s )</td>
<td>System resistance coefficient [s²/m⁵]</td>
</tr>
<tr>
<td>( \eta )</td>
<td>Efficiency [-]</td>
</tr>
<tr>
<td>( \eta_S )</td>
<td>System efficiency [-]</td>
</tr>
<tr>
<td>( \xi )</td>
<td>Axial velocity change ratio [-]</td>
</tr>
<tr>
<td>( \rho )</td>
<td>Fluid density [kg/m³]</td>
</tr>
<tr>
<td>( \sigma )</td>
<td>Solidity [-]</td>
</tr>
<tr>
<td>( \psi_{loss,other} )</td>
<td>Coefficient of the other losses [-]</td>
</tr>
<tr>
<td>( \omega )</td>
<td>Angular speed [rad/s]</td>
</tr>
</tbody>
</table>

Subscripts

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>( cascade )</td>
<td>Related to cascade</td>
</tr>
<tr>
<td>( f )</td>
<td>Front rotor</td>
</tr>
<tr>
<td>( FR )</td>
<td>Only controlling front rotor speed</td>
</tr>
<tr>
<td>( hub )</td>
<td>At hub blade</td>
</tr>
<tr>
<td>( m )</td>
<td>Average of the variables</td>
</tr>
<tr>
<td>( opt )</td>
<td>At the optimum condition</td>
</tr>
<tr>
<td>( ref )</td>
<td>Reference variables</td>
</tr>
<tr>
<td>( RR )</td>
<td>Only controlling rear rotor speed</td>
</tr>
<tr>
<td>( t )</td>
<td>Total of front and rear rotors</td>
</tr>
<tr>
<td>( tip )</td>
<td>At blade tip</td>
</tr>
</tbody>
</table>
2. Performance prediction model

2.1 Overall strategy

Before the detailed description of the performance prediction model, the overall strategy of the performance prediction model is introduced firstly. It has three steps in the performance prediction model as shown in Fig. 1. The 1st step is to determine the theoretical head; the 2nd step is to evaluate the loss quantities; the 3rd step is to predict the head and efficiency.

![Fig. 1 Main components of the performance prediction model](image)

* means base data using CFD under design rotational speed

To construct the performance prediction model along the above procedure, base flow data of test contra-rotating axial flow pump which can be obtained by CFD simulations, are necessary. It should be emphasized that CFD simulations are conducted only for the design rotational speed conditions; such simulations are not additional tasks since they are generally carried out during the usual pump design. Using the base data for the * marked components illustrated in Fig. 1, models of empirical deviation, blade-rows interaction and other losses are constructed. Then, the proposed prediction model is used to predict the performances under various rotational speed conditions without conducting further CFD simulations.

<table>
<thead>
<tr>
<th>Diameter [mm]</th>
<th>RR2 type: $N_{d,f} = N_{d,r} = 1225 \text{min}^{-1}$</th>
<th>RR3 type: $N_{d,f} = 1311 \text{min}^{-1}, N_{d,r} = 1123 \text{min}^{-1}$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Hub</td>
<td>Mid-span</td>
</tr>
<tr>
<td>Front Rotor</td>
<td>100</td>
<td>149</td>
</tr>
<tr>
<td>Solidity $\sigma$ [-]</td>
<td>1.290</td>
<td>0.898</td>
</tr>
<tr>
<td>Stagger Angle $\gamma$ [°]</td>
<td>51.72</td>
<td>68.48</td>
</tr>
<tr>
<td>Rear Rotor</td>
<td>5</td>
<td></td>
</tr>
<tr>
<td>Hydrofoil</td>
<td>NACA4410</td>
<td>NACA4408</td>
</tr>
<tr>
<td>Solidity $\sigma$ [-]</td>
<td>0.840</td>
<td>0.720</td>
</tr>
<tr>
<td>Stagger Angle $\gamma$ [°]</td>
<td>64.24</td>
<td>72.54</td>
</tr>
</tbody>
</table>
2.2 Test rotors

Two previous-designed contra-rotating axial rotors are employed in this study: RR2 type (Shigemitsu et al., 2003) and RR3 type (Cao et al., 2013a). The experimental results of RR2-type rotors will be used to verify the CFD and the proposed performance prediction model, while the RR3-type rotors are employed to demonstrate the energy saving application of the proposed model. Both types of rotors have been designed for the following specifications: total head \( H_{t,d} = 4 \text{ m} \), flow rate \( Q_d = 70 \text{ L/s} \), specific speed of front and rear rotors \( N_{s,f} = N_{s,r} = 1500 \text{ [min}^{-1}, \text{ m}^{3}/\text{min}, \text{ m}] \). The RR2-type rotors are designed with equal speed, in which strong blade-rows interactions and significant cavitation have been observed (Shigemitsu et al., 2003). RR3-type rotors have improved the weaknesses (i.e. strong interaction and remarkable cavitation) of RR2-type rotors by using different-speed design method (Cao et al., 2013a). The main profile information of RR2-type and RR3-type rotors are summarized in Table 1, the shapes of the test rotors are illustrated in Fig. 2. The casing inner diameter is \( D_c = 200 \text{ mm} \), which results in the tip clearance of 1mm.

![Fig. 2 Shapes of test rotors](image)

2.3 CFD analysis for base data

In the present paper, CFD simulations are conducted for the design speed operations to obtain the base flow data which are used to construct the performance prediction model. Then, the CFD simulations are also conducted to validate the model under the rotational speed control (RSC). The both simulations are made by a commercial CFD code: ANSYS CFX 18.0/2019 R3. Figure 3 illustrates CFD models for the numerical simulations. The inlet boundary is located at \( 4D_c \) upstream of the leading edge of front rotor. The outlet boundary is located at \( 1.3D_c \) downstream of the trailing edge of rear rotor or \( 1.7D_c \) downstream of the trailing edge of front rotor. Even though Momosaki et al. (2010b) have well predicted the performance and internal flow in contra-rotating axial flow pump at both of design and off-design conditions by conducting unsteady Reynolds-Averaged Navier-Stokes (RANS) simulations, such computation is too expensive to validate all the results calculated by the performance prediction model. On the other hand, we are focusing on establishing a performance prediction model to evaluate performance under operations with high performance where unfavorable flows may be well relieved, therefore, the steady RANS equations are solved in only one passage of front and rear rotors in contra-rotating axial flow pump, which has good enough accuracy with reasonable time consumption. The Shear Stress Transport (SST) model are employed as a turbulence model in the RANS simulations. The mixing plane is also located between front and rear rotors to help calculate the steady flow through the front and rear rotors. The flow data at the mixing plane will be averaged in circumferential direction on both the outlet of front rotor and inlet of rear rotor, which is like a real mixing process.
2.3.1 CFD numerical models

Computational meshes are generated by using ANSYS TurboGrid 18.0/2019 R3. In order to capture the tip leakage flow, 8 elements are distributed in the blade tip clearance. Since the grid size and normalized distance $y^+$ between first grid layer and boundary have remarkable influence on the CFD based numerical results (Wilson et al., 2001; Hirano et al., 2019), the grid independency of numerical models has been performed. As shown in Table 2 which summarizes the grid-dependence checks for the RR3-type rotors, the performances are evaluated in various grids. It is found that, the cases with nodes over about 1 million and average $y^+$ below about 12.9, from Case 2 to Case 4, show the similar numerical performance results. Therefore, the meshes for all the CFD models are set to be similar to that of the Case 2.

<table>
<thead>
<tr>
<th>Case</th>
<th>Nodes</th>
<th>Average $y^+$ [-]</th>
<th>Total Head $H_t$ [m]</th>
<th>Total Efficiency $\eta_t$ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>344,734</td>
<td>12.79</td>
<td>3.86</td>
<td>0.790</td>
</tr>
<tr>
<td>2</td>
<td>1,036,672</td>
<td>12.93</td>
<td>3.95</td>
<td>0.801</td>
</tr>
<tr>
<td>3</td>
<td>1,227,700</td>
<td>6.65</td>
<td>3.96</td>
<td>0.800</td>
</tr>
<tr>
<td>4</td>
<td>2,792,830</td>
<td>12.96</td>
<td>3.98</td>
<td>0.802</td>
</tr>
</tbody>
</table>

In our previous study, the significant flow interaction has been experimentally observed between the front and rear rotors of a contra-rotating axial flow pump (Cao et al., 2013b). Because of the complexity of blade-rows interaction, in the present study, we have tried to remove the effect of blade-rows interaction in the front rotor performance prediction model, while we have included all such interactions in the rear rotor performance prediction model. As will be seen in the results in Section 3, this strategy is effective and can well help the proposed model predict the total performances. Therefore, as shown in Fig. 3, two types of CFD model are constructed: only front rotor, and the both front and rear rotors. Results of the only-front-rotor CFD model are employed to construct the performance prediction model for front rotor, while those of the both of CFD models (only front rotor, the both front and rear rotors) are applied to establish the performance prediction model for rear rotor.
2.3.2 Accuracy of CFD simulations

Experiments of contra-rotating axial flow pump have been conducted in our previous studies (Shigemitsu et al., 2003; Cao et al., 2013a). The detailed information on the experimental test rig can be found in our previous researches (Furukawa et al., 2007; Shigemitsu et al., 2009). It should be noted that, in the experiments, the performances are evaluated from the casing-average static pressure at Pos. 0 and Pos. 5 (shown in Fig. 4) and the torques of the rotors. For the direct comparison of the performance with experiment, the head in CFD analysis is also evaluated using the casing-average static pressure at the same positions as in the experiment.

Figure 5 displays the performances of experiments and CFD simulations for RR2-type and RR3-type rotors operated in the design rotational speeds. It can be easily found that, significant discrepancies occur in the head evaluations at very low flow rates, which may arise from errors of the steady calculation. Small discrepancies near the design flow rates seem to be the result of over evaluated losses due to the mixing plane (well used in steady calculation) applied between the front and rear rotor domains. Actually, it has been shown that the unsteady simulation of full rotors which can properly take account of rotor-rotor interaction improve the accuracy of performance prediction, realizing much better agreement with experiment (Denton, 2010). Since the discrepancy between CFD and experiment is still small enough and their tendencies in performance change agree well near the design flow rate, CFD results at the near-design flow rates with design rotational speed will be chosen as the base data for the construction of performance prediction model.

2.4 Theoretical head prediction

2.4.1 Basic equations

In order to simplify the flow in the performance prediction model, the following assumptions are employed: negligible viscous losses along the streamtube, axisymmetric flow, no reverse flow, and uniform flow with no swirl at front rotor inlet. Figure 6 illustrates the meridional plane and a typical streamtube in contra-rotating axial flow rotors. At the inlet and outlet of the rotors, a radial equilibrium condition (Brennen, 1994) is applied as follows:
\[
\frac{dp}{dr} = \rho \frac{v_o^2}{r} \tag{1}
\]

where \( p \) denotes the static pressure, \( r \) the local radius, \( \rho \) the fluid density and \( v_o \) the circumferential component of absolute velocity. Since negligible loss is assumed along the streamtube, a rothalpy conservation equation (Inoue and Kamata, 1989) along each streamtube will be:

\[
\frac{p_1}{\rho} + \frac{1}{2} w_1^2 - \frac{1}{2} (r_1 \omega)^2 = \frac{p_2}{\rho} + \frac{1}{2} w_2^2 - \frac{1}{2} (r_2 \omega)^2 \tag{2}
\]

where subscripts 1 and 2 denote the inlet and outlet of rotors respectively, \( w \) represents the relative velocity and \( \omega \) is the rotational speed of the considered rotor. The local mass conservation in the streamtube can be written as follows:

\[
v_{a1} r_1 dr_1 = v_{a2} r_2 dr_2 \tag{3}
\]

where \( v_a \) means the axial velocity. Furthermore, in order to determine the velocities, the exit flow angle \( \beta_2 \) is also necessary and can be expressed as \( \beta_2 = \beta_{b2} + \delta \), where \( \beta_{b2} \) and \( \delta \) are the blade exit angle and the deviation angle respectively. An empirical deviation angle equation (Lieblein, 1965) is introduced here:

\[
\delta = \delta_{ref} + k (i - i_{ref}) \tag{4}
\]

where \( i \) denotes the incidence angle, and \( \delta_{ref} \) and \( i_{ref} \) mean the reference deviation angle and reference incidence angle respectively. The empirical coefficient \( k \) is related to the inlet flow angle \( \beta_1 \) and solidity \( \sigma \), the detail of which can be found in Lieblein’s paper (1965). The reference angles \( \delta_{ref} \) and \( i_{ref} \) are selected as the angles at the design flow rate with the design rotational speed which can be derived from the flow database obtained by CFD.

Together with Eqs. (1)-(4), by using velocity triangles of front and rear rotors, the following differential equation on axial velocity at rotor outlet \( dv_{a2}/dr_2 \) is derived:

\[
dv_{a2} \frac{dr_2}{r_2} = \cos^2 \beta_2 \left\{ 2 \omega \tan \beta_2 - \frac{\tan \beta_2}{\cos^2 \beta_2} \frac{dp}{dr_1} v_{a2} + \frac{1}{r_1} \frac{dv_{a1}}{dr_1} r_2 - \tan \beta_2 \frac{v_{a2}}{r_2} \right\} + \frac{r_2}{v_{a1} r_1} \left[ \frac{dp}{\rho dr_1} + (v_{\theta1} + \omega r_1) \frac{dv_{\theta1}}{dr_1} + \omega v_{\theta1} \right] \tag{5}
\]

This equation is the ordinary differential equation and can be easily solved numerically with sufficient accuracy. The axial velocity is calculated in the 2nd order precision using Taylor’s series, and the other velocities are determined with the velocity triangles at the inlet and outlet of front and rear rotors. Finally, the theoretical head of each rotor can be calculated by the following equation:

\[
H_{th} = \frac{T \omega}{\rho g Q} = \frac{\omega}{g Q} \int_Q (r_2 v_{\theta2} - r_1 v_{\theta1}) dQ \approx \frac{\omega}{g Q} \int_{r_{hub}}^{r_{tip}} (r_2 v_{\theta2} - r_1 v_{\theta1}) 2\pi r_1 v_{a1} dr \tag{6}
\]

Where \( T \) denotes the torque of rotors, \( g \) is the gravitational acceleration, \( r_{tip} \) and \( r_{hub} \) mean the radius at blade tip and the hub respectively.

### 2.4.2 Blade-rows interaction modification

To validate the prediction of theoretical head described above, the theoretical head are evaluated from the mass-averaged total pressure at the rotor-adjacent cross sections \( f_1, f_2 \) and \( f_2 \) (shown in Fig. 4) using the base flow data provided by CFD. Figure 7 (a) illustrates the theoretical head evaluated by CFD and the calculation with above equations near the design flow rates with the design rotational speed. In addition to RR2 type, the prediction of theoretical head (Euler head) of RR3 type is also compared with CFD. In the both two types, good agreement is basically seen in the predictions of all rotors, while, if we look closely at the rear rotor especially for RR2, un-ignorable discrepancies still exist. It should be noted that, the CFD results of front rotors are obtained using the numerical model of only front rotor, while those of rear rotors are obtained using the model considering the both front and rear rotors. Furthermore, significant blade rows interactions have been experimentally observed between the front and rear rotors of contra-rotating axial flow pump in our previous study, especially in the RR2-type rotors (Cao et al., 2013b). Therefore, such discrepancy in the
theoretical head prediction of rear rotor seems to be due to the remarkable blade-rows interactions.

Cao et al. (2013b) have found that the flow field generated by the rear rotor has a significant influence on the flow around the front rotor because of the large stagger angle of rear rotor which is determined considering the exiting swirling flow from the front rotor. The low-pressure region on the suction surface of rear rotor extends into the blade passage of front rotor, which becomes more significant at lower flow rates than at the design one. Furthermore, Zhang et al. (2018) have observed the unsteady vortex behaviors in tip region between front and rear rotors at low flow rates, which is caused by the interaction of backflow from the rear rotor tip with mainstream from the front rotor. Therefore, it seems possible to correlate the lift coefficient of rear rotor blade at tip with the discrepancy of theoretical head prediction, which is expected to help us to improve the theoretical head prediction.

The lift coefficient $C_L$ can be simply derived from the momentum and energy conservation laws of the flows in the cascade, considering the axial velocity change from the inlet to the outlet (Ikui and Inoue, 1988):

$$C_L = \frac{2}{\sigma} \left[ (1 - \frac{\xi}{2}) \tan \beta_1 - (1 + \frac{\xi}{2}) \tan \beta_2 \right] \cos \beta_m - C_D \tan \beta_m$$  \hspace{1cm} (7)

where $\xi$ denotes the axial velocity change ratio defined with the inlet and outlet axial velocities as $\xi = 2 (v_{az} - v_{az})/(v_{az} + v_{az})$, $\beta_m$ is the average flow angle determined from $2 \tan \beta_m = (1 - \xi/2) \tan \beta_1 + (1 + \xi/2) \tan \beta_2$, and $C_D$ is the drag coefficient. The drag coefficient is expressed as follows, using the cascade loss coefficient $\zeta_c$ (introduced by Eq. (9) in Section 2.5.1).

$$C_D = \frac{1}{\sigma} \frac{(1 - \frac{\xi}{2})^2 \zeta_c}{\cos^2 \beta_1} \frac{\cos^2 \beta_m}{\cos^2 \beta_1}$$

Figure 7 (b) shows the normalized discrepancy of the theoretical head prediction plotted against the difference of tip lift coefficient from that at the reference, i.e. at the design flow rate with design rotational speed. The normalized discrepancy $y$ is defined as:

$$y = g \frac{H_{th.r.\text{model}} - H_{th.r.\text{CFD}}}{(\omega_r r_{tip})^2}$$

where $H_{th.r.\text{model}}$ denotes the rear rotor theoretical head calculated by the performance prediction model without considering the blade-rows interactions, $H_{th.r.\text{CFD}}$ is the rear rotor theoretical head calculated by CFD, and $r_{tip}$ means the radius of blade tip. As displayed in Fig. 7 (b), an approximated linear relation can be observed between the normalized discrepancy $y$ and the difference of rear rotor tip lift coefficient as $y = 0.1037 (C_{L.r,tip} - C_{L.r,tip,ref})$ for the both RR2 and RR3 rear rotors. Finally, the predicted theoretical head of rear rotor $H_{th.r}$ with considering blade rows interaction modification will be:
\[ H_{th,r} = H_{th,r,\text{model}} - 0.1037(C_{L,r,\text{tip}} - C_{L,r,\text{tip,ref}}) \left( \frac{\omega r_{\text{tip}}}{g} \right)^2 \]  

(8)

**2.5 Loss models**

We divide the flow losses in the performance prediction model simply into two parts: cascade loss and other losses. The cascade loss is directly evaluated by employing an empirical cascade loss model (Lieblein, 1959). On the other hand, the other losses are modelled according to the blade tip lift coefficient.

**2.5.1 Empirical cascade loss**

According to Lieblein’s paper (1959), the empirical cascade loss coefficient \( \zeta_c \) is given by:

\[
\zeta_c = 2\left( \frac{\delta m^2}{l} \right) \cos^2 \beta_1 \left( \frac{2.16}{\cos^3 \beta_2} \left[ \frac{1.08 \sigma}{2.24} \right] \right) \]  

(9)

where the loss coefficient \( \zeta_c \) is defined with total pressure loss \( p_{\text{loss}} \) and inlet relative velocity \( w_1 \) as \( \zeta_c = 2p_{\text{loss}}/(\rho w_1^2) \). \( (\delta m^2)/l \) denotes momentum thickness coefficient which is calculated by

\[
(\delta m^2)/l = \begin{cases} 
0.004/(1 - 1.17 \ln D_{eq}) & D_{eq} \leq 2 \\
0.004/(1 - 1.17 \ln 2) + 0.11(D_{eq} - 2) & D_{eq} > 2 
\end{cases}
\]

where \( D_{eq} \) is the equivalent diffusion factor. It should be noted that we have added the equation with \( D_{eq} > 2 \) to well achieve loss calculation convergence.

The cascade loss can be locally calculated at every radial location, and then the mass-averaged cascade loss head \( H_{\text{loss,cascade}} \) is determined by:

\[
H_{\text{loss,cascade}} = \frac{1}{2gQ} \int w_1^2 \zeta_c dQ = \frac{1}{2gQ} \int_{r_{\text{hub}}}^{r_{\text{tip}}} w_1^2 \zeta_c 2\pi r v_{\text{at}} dr
\]

(10)

In order to compare with cascade loss model, local losses in CFD simulations are also evaluated using the local theoretical head and the local head rise. Figure 8 shows the local loss distribution of the front rotor predicted by the cascade loss model and the CFD at 90% and 110% of the design flow rate. It is easily found that the loss model well predicts the local loss in the region from the hub to mid-span while the significant discrepancy occurs in the tip region, which may be the result of tip clearance effect.

![Fig. 8 Cascade loss distribution of front rotor evaluated by performance prediction model compared with local flow losses calculated by CFD simulations](image-url)
2.5.2 Empirical other losses

Lakshminarayana (1970) has found that the tip lift coefficient has a strong relation with the losses due to tip clearance effect. It is known that the pressure difference between the pressure and suction surfaces in the blade tip usually causes the leakage flow in the tip clearance. The interactions of tip leakage flow and the mainstream could generate the tip leakage vortex (TLV). As a result, the blockage effect of TLV contributes significantly to loss generation (Denton, 1993; Zhang et al., 2020). It should be noted that the pressure difference in the blade tip can be related to the blade tip lift coefficient. Therefore, in the present paper, we assume the other losses whose dominant component is the loss caused by tip clearance effect, which is herein modelled on the basis of blade tip lift coefficient.

In order to derive the empirical equation for losses except the cascade loss, the total head loss in CFD simulation is evaluated for the both RR2- and RR3-types under the conditions with the design rotational speed near the design flow rate, considering the control volumes defined with f1, f2 and r2 cross sections illustrated in Fig. 4. The total head and the head loss can be obtained by the following equations.

\[ H_{\text{CFD}} = \frac{1}{\rho g} Q \left( \int_{f2 \text{ or } r2} p_t dQ - \int_{f1 \text{ or } f2} p_t dQ \right) \]  

(11)

\[ H_{\text{loss,CFD}} = H_{\text{th,CFD}} - H_{\text{CFD}} \]  

(12)

The quantity of other losses is evaluated from CFD loss \( H_{\text{loss,CFD}} \) by subtracting the cascade loss \( H_{\text{loss,cascade}} \) evaluated by Eq. (10). The other loss coefficient \( \psi_{\text{loss,other}} \) is defined:

\[ \psi_{\text{loss,other}} = \frac{2g}{w_{m,\text{tip}}} (H_{\text{loss,CFD}} - H_{\text{loss,cascade}}) \]  

(13)

where the tip average relative velocity \( w_{m,\text{tip}} \) is determined with the relative velocities at inlet \( w_{1,\text{tip}} \) and outlet \( w_{2,\text{tip}} \) of the blade tip by \( w_{m,\text{tip}} = (w_{1,\text{tip}} + w_{2,\text{tip}})/2 \).

Figure 9 (a) shows the other loss coefficient calculated by Eq. (13) against the tip lift coefficient near the design flow rate with the design rotational speed. It is found that the loss variation is small in the low lift coefficient range, while the rapid change of the loss is found in the range of high lift coefficient. We can also find that the other losses in RR2-type rear rotor has much steeper slope compared with RR3-type rear rotor. Therefore, according to the results under the designed rotational speed, the approximated function for the other loss coefficient in the front rotor of RR2-type, the front and rear rotors of RR3-type is written as:
\[ \psi_{\text{loss,other}} = \begin{cases} 0.0176C_{L,\text{tip}} + 0.0152 & C_{L,\text{tip}} \leq 0.45 \\ 0.16C_{L,\text{tip}} - 0.0489 & C_{L,\text{tip}} > 0.45 \end{cases} \] (14)

For RR2-type rear rotor, the approximated function is written as:

\[ \psi_{\text{loss,other}} = \begin{cases} 0.0536C_{L,\text{tip}} + 0.0187 & C_{L,\text{tip}} \leq 0.75 \\ 0.223C_{L,\text{tip}} - 0.1084 & C_{L,\text{tip}} > 0.75 \end{cases} \] (15)

In Fig. 9 (b), the other loss coefficient in many conditions with the off-design rotational speeds are also plotted against the tip lift coefficient. It can be found that, Eqs. (14) and (15) still well held in the all examined control cases. Finally, the total loss quantity \( H_{\text{loss}} \) is determined with the empirical cascade loss coefficient \( \zeta_c \) and the empirical other loss coefficient \( \psi_{\text{loss,other}} \):

\[
H_{\text{loss}} = \frac{1}{2g} \int w_1^2 \zeta_c dQ + \frac{1}{2g} w_{m,\text{tip}}^2 \psi_{\text{loss,other}}
\] (16)

2.6 Performance predictions

Using the above equations, the theoretical head \( H_{th} \) and the loss quantity \( H_{\text{loss}} \) can be determined. The head \( H \) and the efficiency \( \eta \) can be finally predicted by the following equations.

\[
H = H_{th} - H_{\text{loss}}
\] (17)

\[
\eta = \frac{H}{H_{th}}
\] (18)

In order to examine the accuracy of performance prediction model, CFD simulations are conducted at various flow rates with off-design rotational speeds. Figure 10 displays the predicted head and efficiency of whole rotors as well as those calculated by CFD. It is found that the almost all predictions are located in the range from 90% to 110% of CFD results, indicating the good enough accuracy of performance prediction model.

![Fig. 10 Performances evaluated by performance prediction model and CFD at various flow rates with off-design speeds](image)

(a) Total head

(b) Total efficiency

3. Results and Discussion

3.1 Rotational speed control of each rotor

In our previous study (Momosaki et al., 2010a), rotational speed control (RSC) has been experimentally applied in the front and rear rotors of a contra-rotating axial flow pump with RR2-type rotors. The control information of rotational speed of rotors is illustrated in Fig. 11 (a), where the rotational speed of each rotor is normalized by the designed one, i.e. \( N_d (= N_{d,f}, N_{d,r}) = 1225 \text{ min}^{-1} \). The FR method means only controlling the rotational speed of front rotor, while RR method means only controlling that of rear rotor. At higher flow rates including the designed one (\( Q_d = 70L/s \)), the
internal flow of the front rotor is usually smooth similar to the conventional rotor in rotor-stator type axial flow pump, and therefore only RR method is still effective for the performance improvement. On the other hand, at the low flow rates where the flow recirculation forms at the inlet tip and/or the outlet hub in the front rotor, the performance of the front rotor is significantly deteriorated so that the front rotor speed control (FR) is necessary for the improvement.

The results of performance prediction model under rotational speed controls (FR and RR) are compared with those obtained by experiments and CFD simulations in Fig. 11. In the experiments, the pump performance is evaluated from the measurements of casing-average static pressure and torques of rotors, while the performance prediction model considers the input and output energy of the flow into rotors by using mass-averaged total pressure and mass-averaged theoretical head. The CFD simulation can evaluate the performance in the both experimental and model’s methods. Therefore, we compare the performances evaluated by the experiment and the model prediction in the following way. Figure 11 (b) displays the performances evaluated by the experiments and CFD using the experimental method, while Fig. 11 (c) illustrates the performances evaluated by CFD and predicted by the proposed model in the model’s method. In Fig. 11 (b), fairly good agreement is confirmed in the head and efficiency evaluated by experiment and CFD, suggesting us that the results of CFD are acceptable to be used for the validation of the proposed prediction model. The small discrepancy seems to be due to the limitations of steady one-pitch simulation (mixing plane and steady assumption), which is expected to be minimized by conducting the unsteady simulation considering the full pitch (the whole flow passages) of the both front and rear rotors. In Fig. 11 (c), we can find the negligible discrepancy between the results of CFD and model near the design flow rate, and the small discrepancy is observed only at well off-design flow rates, which means that the proposed model offers good prediction accuracy. However, it should be noted that the proposed model cannot calculate the flows at very low flow rates ($Q < 21 \text{L/s}$ in this case), where the unfavorable back flow phenomena may be unavoidable even with the rotational speed control (RSC).

3.2 Energy saving application
3.2.1 System resistance consideration: problem setting

In Fig. 11, the rotational speed control (RSC) was applied in the RR2-type front and rear rotors without considering the resistance of pump system. Since the pump operation point is determined by the pump head curve and system resistance curve, it is necessary to consider the system resistance curves in actual operations (Turbomachinery Society
of Japan, 1991). The system resistance characteristics $H_R$ is generally expressed by the following equation.

$$H_R = H_0 + \zeta_s Q^2$$  (19)

where $H_0$ is the necessary head of pump which should be specified depending upon the application, $\zeta_s$ denotes the system resistance coefficient and $Q$ means the volumetric flow rate in [m$^3$/s]. In the present paper, two pump system resistances $H_R$ are assumed as follows:

System resistance 1: $H_0 = 0$ m and $\zeta_s = 1200$ s$^2$/m$^5$
System resistance 2: $H_0 = 3$ m and $\zeta_s = 166$ s$^2$/m$^5$

The former corresponds to the case in which the pump is operated in a closed circuit, while the latter does to the case in which the pressurized liquid is necessary, which can be often seen in practical applications.

3.2.2 Optimum operation determination

To maximize the global energy saving for the given system resistances, the input power to the pump should be minimized for the specified flow rate, while keeping the pump head larger than the system resistance head. The input power means the shaft power ($L = \rho g Q H_t h$), therefore the problem is now to minimize the theoretical head $H_t h$ with $H \geq H_R$. The optimum rotational speeds of the front and rear rotors should be determined to satisfy this condition. Since the proposed performance prediction model under rotational speed control (RSC) of rotors is very simple, we can find the optimum speeds easily in the following way.

Firstly, the proposed model is applied to predict the performances within a wide rotational speed range of front and rear rotors in every 20min$^{-1}$ step. Then, according to the system resistances, we select all combinations of the front and rear rotor speeds with high efficiency among the speed combinations with which the pump head satisfies the required resistance head. Finally, the performances are locally approximated with the 2nd order of Taylor’s series by using the model predicted data. The optimum operation points are the conditions satisfying the resistance head with minimum theoretical head at each flow rate in the approximated performances.

![Fig. 12 Rotational speed information for optimum and near-optimum performances for two system resistances](image)

Fig. 12 shows the rotational speed information of the front and rear rotors, which are normalized by the design rotational speed of each rotor. Since the prediction errors may exist in the proposed model, more operations near the optimum performance ($H_{opt} \pm 0.5\%$ and $H_{th,opt} \pm 0.5\%$) are also predicted. Their upper and lower limits of the rotational speeds are plotted with the ‘+’ symbols in Fig. 12. As shown in Fig. 12 (a), the optimal rotational speeds of front and rear rotors linearly decrease with the decrease of flow rate for System resistance 1. The speed ratio of the front and rear rotors is almost constant regardless of flow rate, which implies the flow similarity in the front and rear rotors at each flow rate. It is not surprising since the necessary head rise is $H_0 = 0$ m in System resistance 1 and the resistance head is purely proportional to the flow rate squared; in such case, the control theory should be the same as that of conventional rotor-stator type axial flow pump, and the head coefficients of the rotors can be constant with the maximum
efficiency. In Fig. 12 (b) for System resistance 2, the optimum rotational speeds of front and rear rotors linearly decrease with the increase of flow rate near the design flow rate, whereas the speed ratio of the front and rear rotors is not constant. In addition, at low flow rates, some complex control is necessary to obtain good energy performance. Therefore, it can be mentioned that the proposed model will be very necessary and useful to determine the optimum operation points in those conditions.

In summary, it is found that the favorable operations (including optimum and near-optimum) can be determined by the proposed model in very broad flow rate range for System resistance curve 1, while the favorable operations can only be decided by the proposed model in a limited flow rate range for System resistance curve 2. As mentioned above, the performance prediction model can only be used in the conditions where no reverse flow occurs. The reverse flow may be unavoidable in the conditions with high pressure rise at very low flow rates. Actually, in Fig. 12 (b), there are no plots at the flow rates lower than 35L/s, since it was not possible to determine the optimum rotational speeds by the proposed prediction method. At such flow rates, the low energy performance is unavoidable with the low efficiency operation of the pump and/or the large loss generated by adjusting the valve opening.

3.2.3 Performance prediction and their validations

Figure 13 summarizes the performances evaluated by the proposed model with the optimum rotational speed of rotors achieved by rotational speed control (RSC). CFD simulations have been also conducted to validate the predictions of the model. RSC is applied with maximizing the valve opening as much as possible to reduce the consumed energy there; the equality of pump head with the system resistance indicates that the maximum valve opening is reached. The performances under the design rotational speed (traditional valve control) which are obtained by experiment and CFD are also illustrated to compare with the performances with RSC method. The traditional valve control means that the operational flow rate is adjusted not by RSC but by the opening of valve installed on the pipeline.

In the both system resistance curves 1 and 2, we can observe very small discrepancies in the head and efficiency curves between the CFD and the proposed model under RSC method (red plots), meaning the good prediction accuracy of the proposed model. It is also found that RSC method could well modify the head to satisfy the resistance curve in the wide flow rate range with significantly improved efficiency in the both system resistance cases. This implies that the large amount of energy could be well saved by using the RSC method.
In order to compare the effectiveness of the traditional valve control and RSC methods in terms of energy saving, the system efficiency $\eta_S$ is defined with the system resistance head $H_R$ and theoretical head $H_{th}$ as

$$\eta_S = \frac{H_R}{H_{th}} \quad (20)$$

The system efficiency represents the ratio of system required power and system input power. Therefore, the larger the system efficiency is, the better energy saving will be achieved. Figure 14 shows the system efficiencies under the traditional valve control and RSC methods for the two system resistance curves. As we can see, higher system efficiency can be achieved by RSC method in the wide flow rate range. The traditional valve control shows very low system efficiency at the partial flow rates, while the traditional valve control cannot supply enough head to overcome the system resistances at higher flow rates. Such weaknesses of the traditional valve control method can be improved by applying RSC method. Therefore, the RSC offers significant effectiveness in the energy savings at lower flow rates and can extend operation range at higher flow rates.

4. Conclusion

In the present paper, a simple and fast performance prediction model for contra-rotating axial flow pump has been established. The results evaluated by the proposed model has been compared with experiments and CFD simulations. Then, energy saving applications of the proposed model has also been illustrated. Main findings are summarized as follows:

1) By considering the radial equilibrium condition, the conservations of rothalpy and mass through streamtubes, the empirical deviation angle, the blade-rows-interaction and the empirical loss equations, a simple performance prediction model has been constructed for the contra-rotating axial flow pump to find the effective rotational speed control (RSC);

2) Through the comparisons with experimental and CFD results, the proposed model has been found to have good enough accuracy in predicting performances of contra-rotating axial flow pump under RSC in a broad flow rate range. On the other hand, the proposed model also shows limitations in the conditions with high-pressure rise at very low flow rates. The occurrence of reversed flow may be unavoidable at such flow rates even with RSC.

3) In the energy saving applications of the proposed model, compared with the traditional valve control method, the RSC method optimized by the proposed performance prediction model can well adjust the pump head to satisfy the system resistance curves at wide flow rate range with significantly improved efficiency. Good agreements are obtained between the proposed model and the CFD simulations, showing the effectiveness of the proposed performance prediction model.
References


Stepanoff, A. J., Centrifugal and Axial Flow Pumps (1957), John Wiley and Sons, INC.


