Study on Inlet Flow Field Structure and End-Wall Effect of Axial Flow Pump Impeller under Design Condition

Yanlei Guo 1,2, Congxin Yang 1,2,*, Yan Wang 3, Tianzhi Lv 1,2 and Sen Zhao 1,2

1 School of Energy and Power Engineering, Lanzhou University of Technology, Lanzhou 730050, China; coolantpump@lut.edu.cn (Y.G.); l2432043016@163.com (T.L.); zhaosen0213@163.com (S.Z.)
2 Key Laboratory of Fluid Machinery and Systems, Lanzhou 730050, China
3 Nuclear Power Institute of China, Chengdu 610213, China; wyxs@163.com
* Correspondence: ycxwind@163.com

Abstract: The existing experimental technology cannot accurately and quantitatively measure the flow field structure and the wall boundary layer displacement effect in the axial flow pump. Based on SST k-ω turbulence model, a three-dimensional unsteady numerical simulation of the whole flow field of an axial flow pump was presented at the designed operating point to overcome the weakness of traditional measurement methods in measuring the flow field of the axial flow pump. The flow field structure of the axial flow pump inlet was studied quantitatively and the result was compared with the theoretical design value. It was found that there is an obvious impeller rotation effect and end-wall effect in the flow field of the axial flow pump inlet. The flow field structure of the axial flow pump inlet was studied quantitatively and the result was compared with the theoretical design value. It was found that there is an obvious impeller rotation effect and end-wall effect in the flow field of the axial flow pump inlet. The distribution law of the impeller inlet flow field and the crowding coefficient caused by the wall boundary layer were obtained. The pump inlet measurement point in the experiment and calculation domain inlet in the simulation should be kept at a distance of more than 0.5 Ds away from the impeller inlet to eliminate the influence of the impeller rotation effect. Through contrastive analysis, it was found that there is an obvious difference between the calculated value and the design value of the flow field structure due to the end-wall effect. The crowding coefficient should be taken into account when designing an axial flow pump. This study has certain reference significance for further understanding the flow field structure at the inlet of the axial flow pump impeller and improving the design theory of the axial flow pump.

Keywords: axial flow pump; rotation effect; end-wall effect; boundary layer; axial flow velocity; circumferential velocity component; flow angle

1. Introduction

Pumps are necessary power equipment to maintain the normal operation of modern society. They consume about 10% of the world’s energy. As an important form of the vane pump, the axial flow pump is the best machine to handle the working condition of large flow and low head. It is widely used in national economic fields such as long-distance water transfer, urban water supply and drainage, pumped storage, aerospace and biomedicine.

According to statistics [1], as of 2019, China alone constructed 4713 large (installed capacity greater than 10,000 kW) and medium (installed capacity greater than 10,000 kW) pump stations with an installed capacity of 12 million kW, among which, the axial flow pump accounts for more than 70%. Those small pumping stations with large distribution have greater demand for axial flow pumps [2].

Therefore, improving the efficiency of the axial flow pump and other flow passage components of the system has strong practical significance for energy conservation and emission reduction [3]. The improvement of the axial flow pump performance, however, must depend on the perfection and enrichment of the axial flow pump design theory. In addition, an in-depth study of the internal flow field structure in the axial flow pump and quantitative research on the fluid movement law in the pump are the necessary
conditions for perfecting and enriching the design theory of the axial flow pump. With the development of fluid mechanical testing technology, scholars have carried out a lot of research on the internal flow field structure in the axial flow pumps by using the five-hole probe, high-speed camera, PIV and LDV. The current research is mainly focused on three aspects: pressure pulsation, cavitation and inlet and outlet flow field structure.

On one hand, by monitoring the pressure pulsation at the inlet and outlet of the impeller, the pressure fluctuation law was established [4–7].

On the other hand, by capturing the flow field structure of cavitation induced by inlet inflow conditions or tip clearance leakage flow with a high-speed camera, the mechanism of cavitation initiation, development and evolution in the axial flow pump was established [8–10].

The research also aimed to study the flow field structure at the inlet and outlet of the axial flow pump. The earliest literature on the measurement of flow field structure of axial flow pumps was found in NASA’s relevant research on axial flow fuel transfer pumps with high hub ratio and low specific speed in the 1960s. The flow coefficient, liquid flow angle and head coefficient at different radii of the impeller inlet and outlet were measured in detail [11–16]. Zhang et al. [17] studied the distribution law of the axial velocity and circulation at the impeller outlet of a series of high-efficiency axial flow pumps by using a five-hole probe. The measured axial velocity and circulation distribution data were fitted into a polynomial mathematical model, which can provide a reference for the hydraulic design of the axial flow pump impeller. Tang et al. [18] researched the inlet and outlet flow field structure of an axial flow pump impeller under different working conditions by a five-hole probe and PIV technology, respectively. The experimental results were in good agreement with the numerical simulation results under the design condition and large flow condition with the numerical simulation results, but very different under small flow condition. Huang [19] used 2D-PIV technology to study the flow field structure of an axial flow pump. The result showed that the two-dimensional velocity fields at the inlet and outlet of the impeller at different blade height regions were successfully obtained and the separated flow at the trailing edge of the blade and the structure of reflux and vortex at the blade root were observed. Geng [20] measured the impeller outlet flow field of an axial flow pump comprehensively under design condition, small flow condition and large flow condition by using 3D-PIV technology, which is able to study the outlet flow field of the axial flow pump impeller. Then, Zhang et al. [21,22] measured the impeller inlet flow field of an axial flow pump under design condition, small flow condition and large flow condition by 3D-PIV technology and the distribution law of the impeller inlet flow field under different working conditions were obtained. By using numerical simulation and PIV technology, Yang [23] measured the velocity circulation of the inlet and outlet surface of the structural change section of the straight pipe outlet channel under different working conditions and obtained the variation law of the velocity and flow line of the characteristic section. The results also proved that the numerical simulation results were in good agreement with the experimental results. Yangzhou university [24–26] studied the whole flow field of an axial flow pump from the impeller inlet to the guide vane outlet by LDV.

The inlet flow field structure of axial flow pumps is an important factor affecting the hydraulic performance of the axial flow pump, such as efficiency and cavitation. As mentioned above, a large number of experimental studies have been carried out on the inlet flow field structure of axial flow pumps. However, on one hand, most of the test results were not compared with the design values of flow field structure, so these work results were not fed back to the axial flow pump design theory and nor were they used to improve the design theory. On the other hand, the existing test technology could not penetrate accurately into the flow field under the wall boundary layer without damaging the flow field structure.

With the development of computer science, CFD has gradually become the third most important method to study fluid machinery besides theory and experiment since the 21st century. Its accuracy has been proved by a large number of effective applications in the industrial field. It makes it possible to study the flow field structure of the axial flow...
pump quantitatively by using the high-precision numerical calculation method [27–29]. Zhang et al. [30] simulated a hydraulic axial-flow pump model with different tip clearance based on the CFD technology and high-quality structured grids and discussed the influence of tip clearance on the meridional velocity and circulation distributions in impeller inlet and outlet. Feng et al. [31] studied the cavitation performance of an axial flow pump with inlet guide vanes for different flow rates with CFD, which obtained the influence law of the inlet guide vanes on pump hydraulic performance and cavitation and tried to find out the methods to improve the cavitation performance at off-design flow conditions by adjusting angles of inlet guide vanes to positive values at low flow rates and by regulating angles of inlet guide vanes to negative values at high flow rates. Mohammadi et al. [32] calculated the transient inlet flow rate of an axial flow pump as a part of the solution rather than specifying it as a fixed boundary condition with the method of large eddy simulation for the first time. The method serves as a numerical framework for simulating problems incorporating complex shapes with moving/stationary parts at all regimes including the transient start-up and shut-down phases. Tao et al. [33] studied the application of swept blades for axial flow liquid pumps based on the detached eddy simulation turbulence model. The results showed how the swept stacking changes the radial equilibrium of the high density, high viscosity flow and the effects on the mass transfer and pressure pulsations. The swept blade effectively improves the operating stability of high-speed fuel pumps.

Based on SST K-ω low Reynolds number turbulence model, this study presented the unsteady numerical simulation of the whole flow field of an axial flow pump designed by the circular arc method and thickened with 791 airfoil change law. The flow field structure at the impeller inlet was analyzed quantitatively and the result was compared with the theoretical design value, which could provide a certain reference for the enrichment of the design theory of the axial flow pump.

2. Research Project and Method

2.1. Pump Model

The circular arc method takes a circular arc as the pressure surface of the airfoil and thickens from it to the suction surface based on a certain law. The calculation of this method is simple, which makes it the most widely used method in the design of the axial flow pump. The schematic diagram of the airfoil designed by circular arc method is shown in Figure 1.

![Figure 1. Schematic diagram of airfoil designed by circular arc method.](image)

In this study, the circular arc method was adopted for the hydraulic design of an axial-flow pump based on the variation law of the 791 airfoil. The circular arc of each section was thickened from the pressure surface to the suction surface. To facilitate the study of the difference between the design value and the calculated value of the flow field structure in the axial flow pump, zero-incidence was adopted, which means the setting angle at the impeller inlet is equal to the liquid flow angle.

The basic design parameters of the pump are shown in Tables 1–3. The calculation method and selection principle of design parameters are derived from the book *Axial flow pump and mixed flow pump* [34].
Table 1. Pump design parameters.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Sign</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotating Speed</td>
<td>( n )</td>
<td>1450 rpm</td>
</tr>
<tr>
<td>Design Flow Coefficient</td>
<td>( \lambda_{DN} )</td>
<td>0.289</td>
</tr>
<tr>
<td>Design Head Coefficient</td>
<td>( \psi_{DN} )</td>
<td>0.1265</td>
</tr>
<tr>
<td>Impeller Blade Number</td>
<td>( Z_{imp} )</td>
<td>4</td>
</tr>
<tr>
<td>Guide Vane Number</td>
<td>( Z_{gui} )</td>
<td>7</td>
</tr>
<tr>
<td>Support Blade Number</td>
<td>( Z_{sup} )</td>
<td>4</td>
</tr>
<tr>
<td>Tip Clearance</td>
<td>( c )</td>
<td>0.0002 m</td>
</tr>
<tr>
<td>hub-tip radius ratio</td>
<td>( \varepsilon )</td>
<td>0.5</td>
</tr>
<tr>
<td>Impeller Radius</td>
<td>( R_i )</td>
<td>0.15 m</td>
</tr>
</tbody>
</table>

Table 2. The design parameters of impeller.

<table>
<thead>
<tr>
<th>Radial Coefficient ( R^* )</th>
<th>Inlet Setting Angle ( \kappa_1 )</th>
<th>Outlet Crowding Coefficient ( \xi_2 )</th>
<th>Outlet Setting Angle ( \kappa_2 )</th>
<th>Blade Setting Angle ( \gamma )</th>
<th>Cascade Density ( \sigma )</th>
<th>Maximum Thickness Coefficient ( \delta_{max}/L )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>31.4</td>
<td>0.888</td>
<td>58.8</td>
<td>45.1</td>
<td>1.08</td>
<td>0.110</td>
</tr>
<tr>
<td>0.25</td>
<td>25.9</td>
<td>0.900</td>
<td>40.4</td>
<td>33.1</td>
<td>1.02</td>
<td>0.080</td>
</tr>
<tr>
<td>0.5</td>
<td>22.0</td>
<td>0.914</td>
<td>30.4</td>
<td>26.2</td>
<td>0.96</td>
<td>0.059</td>
</tr>
<tr>
<td>0.75</td>
<td>19.0</td>
<td>0.930</td>
<td>24.3</td>
<td>21.7</td>
<td>0.88</td>
<td>0.044</td>
</tr>
<tr>
<td>1</td>
<td>16.7</td>
<td>0.946</td>
<td>20.2</td>
<td>18.4</td>
<td>0.8</td>
<td>0.032</td>
</tr>
</tbody>
</table>

Table 3. The design parameters of guide vane.

<table>
<thead>
<tr>
<th>Radial Coefficient ( R^* )</th>
<th>Inlet Setting Angle ( \kappa_3 )</th>
<th>Outlet Setting Angle ( \kappa_4 )</th>
<th>Blade Setting Angle ( \gamma )</th>
<th>Cascade Density ( \sigma )</th>
<th>Maximum Thickness Coefficient ( \delta_{max}/L )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>48.2</td>
<td>90</td>
<td>69.1</td>
<td>1.76</td>
<td>0.051</td>
</tr>
<tr>
<td>0.25</td>
<td>54.1</td>
<td>90</td>
<td>72.0</td>
<td>1.43</td>
<td>0.050</td>
</tr>
<tr>
<td>0.5</td>
<td>58.5</td>
<td>90</td>
<td>74.2</td>
<td>1.21</td>
<td>0.049</td>
</tr>
<tr>
<td>0.75</td>
<td>61.9</td>
<td>90</td>
<td>75.9</td>
<td>1.06</td>
<td>0.048</td>
</tr>
<tr>
<td>1</td>
<td>64.5</td>
<td>90</td>
<td>77.3</td>
<td>0.95</td>
<td>0.047</td>
</tr>
</tbody>
</table>

The axial flow pump model is shown in Figure 2. Fluid flows through a suction chamber, an impeller, a guide vane, a support and a pressurized chamber.

Figure 2. Axial flow pump model.
2.2. Governing Equations

The fluid motion in the axial flow pump is dominated by three governing equations: mass conservation equation, momentum conservation equation and energy conservation equation. Since the temperature of the transport medium water in the axial flow pump could be regarded as constant in the whole flow process, it could be considered incompressible. Heat transfer can be ignored to save calculation time. Then, the tensor forms of the mass conservation equation and momentum conservation equation of the viscous incompressible medium are shown in Equations (1) and (2).

\[
\frac{\partial u_i}{\partial x_i} = 0 \tag{1}
\]

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = F_i - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} \tag{2}
\]

where
- \(u_i\) is the velocity component in three directions under the rectangular coordinate system,
- \(F_i\) is the mass force,
- \(p\) is the pressure,
- \(\rho\) is the medium density,
- \(\nu\) is the medium kinematic viscosity.

SST (shear-stress-transport) \(k-\omega\) turbulence model, a shear stress transport model, was proposed by Menter [35] based on the modified eddy viscosity coefficient in the turbulence model of BSL \(k-\omega\). SST turbulence model combines the stability near the wall of the model \(k-\omega\) and the independence outside the boundary layer of the model \(k-\epsilon\), which fully considers the transport of turbulent shear stress and could predict accurately the starting position and size of flow separation area under the inverse pressure gradient. The basic equations are as follows:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \frac{\partial k}{\partial x_j} \right] + \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho \omega^2 \tag{3}
\]

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho u_i \omega)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \frac{\partial \omega}{\partial x_j} \right] + \frac{\gamma_1}{v_i} \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta \rho \omega^2 + 2(1 - F_1) \rho \sigma \omega^2 \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \tag{4}
\]

where
- \(k\) is the turbulent kinetic energy,
- \(\rho\) is the medium density,
- \(\mu\) is the kinematic viscosity,
- \(\tau_{ij}\) is the shear stress,
- \(\omega\) is the turbulence frequency,
- \(F_1\) is the mixing function.

The rest are constants: \(\sigma_k = 1.176\), \(\sigma_\omega = 2\), \(\beta = 0.075\), \(\beta^* = 0.09\), \(\kappa = 0.41\), \(\gamma_1 = \beta / \beta^* - \sigma_\omega \kappa^2 / \sqrt{\beta^*}\).

Among them, the mixing function is very important for the accuracy of calculation. The equation is based on the first layer grid height near the wall and related flow variables.

\[
F_1 = \tanh(\text{arg}_1^4) \tag{5}
\]

\[
\text{arg}_1 = \min(\max(\sqrt{\frac{k}{\beta^* \omega y}}, \frac{500 \nu}{\omega y^3 \sigma}, \frac{4 \rho k}{CD \sigma_\omega^2 y^2}), 1) \tag{6}
\]

\[
CD = \max(2\rho \frac{1}{\sigma_\omega^2 \nu} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 1.0 \times 10^{-10}) \tag{7}
\]

where, \(y\) is the distance from the first layer of grid to the wall, \(\sigma_\omega^2 = 1.168\).
The limiting equation of eddy viscosity is as follows:

$$v_1 = \frac{a_1 k}{\max(a_1 \omega, SF_2)}$$

$$v_1 = \mu_1 / \rho$$

where

\( \mu_1 \) is turbulent viscosity,
\( S \) is an invariant measure of strain rate,
\( F_2 \) is a mixed function, which strictly controls the eddy viscosity limit equation in the boundary layer

and \( a_1 \) is a constant, equal to 5/9.

\[
F_2 = \tanh(\arg_2^2) \\
\arg_2 = \max \left\{ 2 \frac{\sqrt{x}}{\omega_{max}/\sqrt{\gamma \mu}}; \frac{500 \nu}{\gamma \mu} \right\}
\]

2.3. Computational Setup

The region near the wall of the boundary layer can be divided into three parts, the viscous sublayer, the buffer layer and the log-law region by velocity gradient as shown in Figure 3. The low Reynolds number turbulence model, SST \( k-\omega \), has extremely strict requirements for grids near the wall. The \( y+ \) needs to be smaller than 1 to ensure the first layer of the grids falls within the viscous sublayer. However, the blade shape of the axial flow pump is twisted and irregular, which makes it very difficult for all grids near the wall to meet the requirement that \( y+ \leq 1 \). The difficulties are mainly reflected in two aspects. On one hand, it will lead to a sharp decline in grids quality. On the other hand, the number of grids will increase sharply on the condition of ensuring the growth rate of grids near the wall, which will lead to a significant increase in computing costs. Relevant research showed that when \( y+ \leq 5 \), the first layer of the grid near the wall is within the viscous sublayer. Therefore, the target \( y+ \) was determined to be within 5 in this study. The final average \( y+ \) of all the grids is 1.1, which meets basically the requirements of SST \( k-\omega \) turbulence model for \( y+ \) as shown in Figure 4.

![Figure 3](image1)

Figure 3. Boundary layer wall ratio.

The grids of the flow domain were generated by the commercial software ICEM CFD. The suction chamber, impeller, guide vane, support and pressurized chamber were meshed by using hexahedral elements. The grid independence was checked by increasing the number of elements and checking that the residual of the hydraulic efficiency changed by less than 0.5% when the number of elements was increased by 10%, which was shown in Figure 5. In addition, 40 layers of grids were set at the tip clearance to capture the flow field structure at the tip clearance. The growth rate of the boundary layer grids was controlled by
1.2. At the same time, this study adopted the processing method of completely consistent correspondence between interface grids nodes to eliminate the data transmission error caused by the inconsistency of interface grids between the rotor domain and stator domain. The final grids were summarized in Table 4 and illustrated in Figure 6. The final grids number is much larger than that required by the grid independence.

![Figure 4](image1.png)

**Figure 4.** The $y^+$ distribution of grids near the wall of impeller and guide vane.

![Figure 5](image2.png)

**Figure 5.** Results of grid independence.

**Table 4.** Mesh details for each domain.

<table>
<thead>
<tr>
<th>Domain</th>
<th>Nodes/Million</th>
</tr>
</thead>
<tbody>
<tr>
<td>Suction Chamber</td>
<td>0.87</td>
</tr>
<tr>
<td>Impeller</td>
<td>14.98</td>
</tr>
<tr>
<td>Guide Vane</td>
<td>15.98</td>
</tr>
<tr>
<td>Support</td>
<td>2.62</td>
</tr>
<tr>
<td>Pressurized Chamber</td>
<td>1.51</td>
</tr>
</tbody>
</table>

The commercial computational fluid dynamics code ANSYS-CFX 20.1 was used to simulate the three-dimensional flow in the pump. The fluid medium was set as liquid water with a density of 998.2 kg/m$^3$ and a dynamic viscosity of 0.000889 Pa·s. The impeller was set as a rotating domain and the others were stationary. The simulation was performed with the moving reference frame model. The relative pressure and the turbulent intensity of the inlet boundary were set as 1 atm and 5%, respectively. The mass flow rate (350 kg/s) was applied for the inlet boundary condition at the suction chamber inflow. A pressure outlet type boundary was set at the pressurized chamber outflow, and all the solid walls were set as no-slip wall boundaries. The length of the suction chamber was 8 $R_t$, and it was 6 $R_t$ long from the end of the diffusion section to the outlet. General grids interfaces were used to connect the rotating and stationary parts with Frozen Rotor at the steady condition and Transient Rotor Stator interfaces at the unsteady condition. In addition, the steady calculation result was used as the initial condition of the unsteady calculation.
the time-step size was set as 0.000115 s to achieve the adequate resolution of simulation results [36], which means, every 1 degree of impeller rotation was a time step. Besides, the convergence precision was set as 0.00001. The steady-state calculation was proposed to accelerate the convergence rate, while its results were taken as the initial condition of the transient numerical simulation.

![Figure 6. Flow domains and grids.](image)

### 2.4. Experimental Setup

An experimental study on the external characteristics of the model pump was carried out to verify the accuracy of numerical simulation. The test rig is shown schematically in Figure 7 and the relevant measuring instruments in Figure 8. A flowmeter, with an uncertainty of less than 1.5%, was used to measure the flow rate \(Q_v\). A torque speed sensor, with speed uncertainty of less than 0.2% and torque uncertainty of less than 1.5%, was used to test the rotation speed and torque. The head \(H\) between the pump inlet and outlet was also measured with pressure sensors with an uncertainty of less than 1%. All the data were converted to digital form and stored in a computer database.

![Figure 7. Schematic map of the hydraulic test rig.](image)
prediction due to the obvious stall, secondary flow and other complex vortex motion in
value of external characteristics near the design working point of the axial flow pump
Therefore, to study the inlet flow field structure. The physical parameters on the intersection lines
used
used
Figure 3.1. Effect of Impeller Rotation on Flow Field Structure
3. Results and Discussion
3.1. External Characteristic Analysis
Figure 9 shows the head coefficient curve and efficiency curve obtained from CFD and test results. The curves of CFD are in good agreement with the curves of the test at the flow coefficient from 0.225 to 0.325. If it deviates too much from the designed working condition, especially under small working condition, CFD cannot give accurate prediction due to the obvious stall, secondary flow and other complex vortex motion in the pump. Therefore, the turbulence model SST k-ω can give a more accurate prediction value of external characteristics near the design working point of the axial flow pump and can be used to study the fluid movement law in the axial flow pump near the design working condition.

Figure 9. External characteristic analysis.

3.2. Internal Characteristic Analysis
3.2.1. Effect of Impeller Rotation on Flow Field Structure
Seven cylindrical layers with radial coefficients of 0.01, 0.05, 0.25, 0.5, 0.75, 0.95 and 0.99 were named as 0, 1, 2, 3, 4, 5, 6 and 7, respectively, and five planes, 0.12 D_S, 0.24 D_S, 0.36 D_S and 0.48 D_S away from the impeller inlet edge were named as A, B, C, D and E to study the inlet flow field structure. The physical parameters on the intersection lines
between the cylinder plane and the plane with different radial coefficients were the research object as shown in Figure 10.

![Figure 10. Schematic diagram of data sampling location.](image)

The numerical study predicted the pressure coefficient distribution of the impeller inlet flow structure at the design condition. Figure 11 shows the pressure coefficient distribution curve on the circumference at different distances from the blade inlet with a radial coefficient of 0.5. A large number of studies have shown that there is an obvious periodic pressure pulsation at the impeller inlet due to the impeller rotation effect. However, it is easy to notice that the closer to the impeller inlet, the more obvious the pressure pulsation is. With the distance away from the impeller inlet, the amplitude of the pressure pulsation decreases rapidly. The pressure pulsation basically disappears when it is greater than 0.5 \( D_S \). Therefore, the inlet boundary of the calculation domain should be kept at a sufficient distance from the impeller domain to ensure the full development of inlet flow and eliminate the influence of ideal inlet boundary condition on the flow field structure. The distance between the inlet of the calculation domain and the inlet of the impeller should not be less than 0.5 \( D_S \).

![Figure 11. Distribution of impeller inlet pressure coefficient at radial coefficient of 0.5.](image)

Figure 11 shows the pressure coefficient \( (C_p = (p - \bar{p}_{in}) / (0.5 \rho V^2_{in})) \) distribution curve in the circumferential direction at different radius coefficients on plane A. It can be seen that the larger the radius coefficient is, the greater the fluctuation amplitude of the pressure coefficient is. The minimum value of the pressure coefficient at the flange position is smaller than that at other positions, which is also the reason why cavitation always occurs first at the blade inlet flange position. The pressure energy and kinetic energy of the incompressible fluid can be transformed into each other along the streamline according to the Bernoulli equation. Therefore, considering the hydraulic loss from the pump inlet to the impeller inlet, the peak value of the impeller inlet pressure coefficient should be less than 1, which is inconsistent with the results shown in Figure 11. It can be seen from Figure 12 that the peak value of the pressure coefficient at each radius coefficient position is much greater than 1. In view of the above, it can be inferred that the impeller rotation effect produces
an unsteady periodic disturbance to the inlet flow field structure and destroys the steady condition on which the Bernoulli equation in the static coordinate depends.

Figure 12. Distribution of impeller inlet pressure coefficient at different radial coefficient in plane A.

The energy head coefficients are defined as follows to represent visually the energy change at the impeller inlet to a higher degree.

\[
\phi_p = \frac{P}{P_{tot-in}}, \quad \phi_d = \frac{0.5 \rho V^2}{P_{tot-in}}, \quad \phi_{tot} = \phi_p + \phi_d
\]

where

- \( P_{tot-in} \) is the total pressure at the pump inlet,
- \( \phi_p \) is the Pressure head energy coefficient,
- \( \phi_d \) is the velocity energy head coefficient,
- \( \phi_{tot} \) is the total energy head coefficient.

Figure 13 shows the distribution of the energy head coefficients in the circumferential direction at the A3 position. It can be seen that the total energy head coefficient presents a periodic distribution in the circumferential direction (if the Bernoulli equation applies here, the total energy head coefficient should be a constant less than 1), and the pressure head and velocity head do not show a relationship of that as one falls, another rises. The rotating effect of the impeller destroys the steady flow condition under which the Bernoulli equation can be established in the static coordinate system.

Figure 13. Distribution of impeller inlet energy head coefficient at different radial coefficient at A3.

Thus, the inferences above can be confirmed. The impeller rotation effect makes the impeller inlet flow field structure present an unsteady periodic distribution and the steady
condition in the static coordinate is damaged so that the Bernoulli equation is no longer applicable here in the static coordinate system.

\[ H = \frac{(\overline{P}_{\text{out}} - \overline{P}_{\text{in}})}{\rho g} + \frac{|(Q_0/A_{\text{out}})|^2 - (Q_0/A_{\text{in}})^2/2g}{\rho g} \]

where, \( \overline{P}_{\text{out}} \) and \( \overline{P}_{\text{in}} \) are the average pressure at the measuring point of the pump inlet and outlet pressure gauge, respectively. \( A_{\text{in}} \) and \( A_{\text{out}} \) are the flow area where the pump inlet and outlet measuring point are located.

We have known that the impeller rotation effect would produce an obvious unsteady disturbance in the impeller inlet flow field, however, it is still unknown if the physical parameters of the impeller inlet flow field based on spatial average are also affected by the impeller rotation effect. It has an important impact on the pump inlet pressure \( \overline{P}_{\text{in}} \) in the experiment if the pump inlet measurement point is just located in the disturbed area.

Therefore, a numerical simulation control group was established, that is, the numerical simulation calculation was carried out in the flow field at the pump inlet under the condition that there is no impeller rotation domain based on the same grid model. Then, the weighted average pressure coefficient based on area from the water guide cone inlet to the plane where the impeller inlet edge is located was calculated. The curves of the weighted average value based on area varying within the impeller inlet edge distance in the two cases are shown in Figure 14. The change curve of the pressure coefficient based on the weighted average area basically coincides in the two cases when the distance is greater than 0.13 \( D_s \), while the pressure coefficient in the case of an impeller existing drops rapidly when the distance is less than 1.3 \( D_s \), and the closer it is to the inlet edge of the impeller, the greater the decline rate. This further proves that, on average, the impeller inlet flow field structure is also affected by the impeller rotation effect. It causes a significant drop in the fluid pressure before the fluid enters the impeller channel. Cavitation may occur before the fluid enters the impeller.

![Figure 14. Distribution of \( C_p \) with distance from impeller inlet.](image)

If the pump inlet pressure measuring point is located in the area affected by the impeller rotation, the experimental head will be larger than the true value. Combined with the conclusion in Figure 11, the measuring point at the pump inlet shall be set at a distance not less than 0.5 times \( D_s \) to ensure the accuracy of the experiment.

### 3.2.2. End-Wall Effect

The cylinder independence assumption is one of the basic assumptions of the axial flow pump design theory. It means the flow between each radial coefficient is independent, and there is no radial flow. However, the existence of no-slip wall such as impeller shell and hub will inevitably affect the impeller inlet flow structure and undermine the assumption of cylinder independence. It is called the end-wall effect. The influence of the end-wall
effect on the structure of flow field should be taken into account in the design of the axial flow pump.

Through the above analysis, we have known that the impeller inlet flow field structure presents an unsteady periodic distribution in the circumferential direction due to the impeller rotation effect. However, for the design of the axial flow pump, the design value can only be the average value of the flow field structure. The weighted average based on mass flow of the physical parameters at each radial coefficient position were took to study the influence of the end-wall effect on the flow field structure.

\[ x_{MCA} = \frac{\sum (\Delta m \cdot x)}{m} \]

where, \( x \) is the physical variable.

Figure 15 shows the distribution of the axial velocity coefficient and circumferential velocity component coefficient along the circumferential direction at the A3 position of the impeller inlet. The axial velocity coefficient and circumferential velocity component coefficient are affected by the impeller rotation effect and distributed periodically in the circumferential direction as same as the distribution of pressure coefficient. The non-pre-rotation inlet was adopted in this study. This means that the circumferential velocity component is equal to 0 m/s, while, in terms of local flow field structure, the circumferential velocity component coefficient at each position is not equal to 0 but fluctuates periodically at the center of 0. It proves that the hydraulic design of axial flow pumps based on the average value is reasonable.

![Figure 15. The velocity coefficient distribution along the circumference at A3.](image)

Figure 16 shows the distribution of the weighted average axial velocity coefficient \( (\varphi_{m1-MCA-CFD}) \) and circumferential velocity component coefficient \( (\varphi_{\mu1-MCA-CFD}) \) based on mass flow with radial coefficient at plane A. The calculated value, \( \varphi_{\mu1-MCA-CFD} \), is basically consistent with the design value, \( \varphi_{\mu1-DN} \), except at the area near the hub wall, which indicates that the impeller inlet of the axial flow pump, on the whole, meets the design condition of non-pre-rotation inlet. In the region near the hub wall, \( \varphi_{\mu1-MCA-CFD} \) increases sharply and reaches the maximum at the hub wall, the reason of which is the existence of boundary layer. The rotation of the impeller hub drives the fluid near the wall to rotate, so that it has a larger circumferential velocity component. The closer it is to the wall, the greater the circumferential velocity component.

The distribution of \( \varphi_{m1-MCA-CFD} \) with the radial coefficient presents a curve which is flat in the main flow region and steep in the region near the wall and \( \varphi_{m1-MCA-CFD} \) in the main flow region is slightly larger than \( \varphi_{m1-DN} \). Similarly, due to the influence of the no-slip wall, there is a boundary layer area with extremely thin thickness near the wall where a large axial velocity gradient exists. The closer to the wall, the smaller the axial velocity is. The existence of the boundary layer will undoubtedly reduce the effective flow. 
cross-section area at the impeller inlet and then increase the flow velocity in the main flow area. That is why \( \varphi_{m1-MCA-CFD} \) in the main flow region is slightly larger than \( \varphi_{m1-DN} \). The velocity coefficient of the plane \( A \) was averaged based on the mass flow weight to study the crowding effect of the boundary layer quantitatively.

\[
\varphi_{m1-MPA-CFD} = \frac{massFlowAve(\varphi_{m1})}{Q_0\rho} = \frac{\sum(\Delta m \varphi_{m1})}{Q_0\rho} = 0.2986
\]

The design value of the axial velocity coefficient is obtained by the following formula.

\[
\varphi_{m1-DN} = \varphi_{m1-APA} = \frac{AreaAve(\Delta A \varphi_{m1})}{A_1} = \frac{Q_v}{A_1 U_t} = 0.289
\]

Then, the crowding coefficient at the impeller inlet is as follows:

\[
\zeta_1 = \frac{\varphi_{m1-DN}}{\varphi_{m1-MPA-CFD}} = 0.968
\]

It can be seen from the above that the boundary layers near the wall of the hub and flange at the inlet of the impeller cause obvious crowding of the flow area and the crowding coefficient is about 0.97. The axial velocity in the mainstream region represents the real axial velocity at the impeller inlet. Therefore, it is necessary to take the weighted average axial velocity based on mass flow rather than the value based on area as the design value. The design value of the axial velocity coefficient is obtained by the following formula.

\[
\varphi_{m1-DN} = \varphi_{m1-MPA} = \frac{\varphi_{m1-APA}}{0.97} = \frac{Q_v}{0.97 A U_t}
\]

Figure 16 has shown that the circumferential velocity component near the wall of the impeller inlet hub increases rapidly, while the axial velocity decreases rapidly, so the absolute velocity in this area will be dominated by the circumferential velocity component as shown in Figure 17. It will result in the increase of the relative flow angle and the decrease of the absolute flow angle at the hub of the impeller inlet. Thus, the flow angle near the hub of the impeller inlet would deviate from the design value. On the other hand, the attenuation of the axial velocity near the flange wall will reduce the relative flow angle in this region but will not affect the absolute flow angle. Figure 18 shows the velocity triangle distribution of the impeller inlet flow field structure. The red is the velocity triangle near the rim, the purple is the velocity triangle near the hub and the black is the design velocity triangle.

![Figure 16. Distribution of impeller inlet velocity coefficient.](image-url)
The design value and $\beta$ of this distribution are quantitatively inferred above, the flow angle near the wall is obviously affected by the wall boundary layer. The relative flow angle based on the weighted average of mass flow, $\beta_{1-MCA-CFD}$, increases rapidly and the absolute flow angle based on the weighted average of mass flow, $\alpha_{1-MCA-CFD}$, decreases sharply at the hub of the impeller inlet, while $\beta_{1-MCA-CFD}$ decreases rapidly at the flange of the impeller. In addition, $\alpha_{1-MCA-CFD}$ except in the hub area is perfectly consistent with the design value and $\beta_{1-MCA-CFD}$ in the main region ($R^* \in (0.05, 0.95)$) is slightly larger than the design value. The reason why $\beta_{1-MCA-CFD}$ is slightly larger than $\beta_{1-DN}$ is that the existence of the boundary layers near the wall of the hub and flange increase the axial velocity in the main region, which has been explained above.

![Figure 17](image1.png)

**Figure 17.** The absolute velocity vector distribution near the impeller inlet hub wall.

![Figure 18](image2.png)

**Figure 18.** The velocity triangle of the impeller inlet flow field structure.

![Figure 19](image3.png)

**Figure 19.** Distribution of liquid flow angle at impeller inlet with radial coefficient.
4. Conclusions

In this study, we made a three-dimensional unsteady numerical simulation of the whole flow field of an axial flow pump at the design operating point. The rotation effect of the impeller and end-wall effect at the axial flow pump impeller inlet were quantitatively studied. The main conclusions are as follows:

1. The impeller rotation produces an obvious unsteady periodic disturbance effect on the impeller inlet flow field structure and the closer to the impeller inlet, the more obvious the disturbance effect is. The disturbance weakens rapidly with the increase of the distance from the impeller inlet and basically disappears when it is greater than 0.5 times \(D_s\). At the same time, the average pressure of fluid drops rapidly in a very small area of 0.13 times \(D_s\) from the impeller inlet. The location of the pressure measuring point at the pump inlet in the test and the location of the inlet boundary of the calculation domain should be kept at a distance of more than 0.5 \(D_s\) away from the impeller inlet to ensure the accuracy of the test and numerical calculation.

2. The Bernoulli equation in the static coordinate system is not applicable in the region of the impeller rotation disturbance.

3. There is an obvious end-wall effect at the impeller inlet of the axial flow pump. The existence of the boundary layer near the hub and flange cause obvious crowding out of the flow channel, which results in the increase of the axial velocity in the main flow region. The crowding coefficient is approximately equal to 0.97. It should be taken into account when designing the axial flow pump.

4. The rotation of the hub makes the fluid in this area have an obvious circumferential velocity component, and the fluid flow angle at this position will deviate far from the design value.

Author Contributions: Conceptualization, C.Y.; software, Y.G., S.Z., and T.L.; validation, Y.G.; formal analysis, C.Y. and Y.W.; investigation, Y.G.; data curation, Y.G.; writing—original draft preparation, Y.G. writing—review and editing, Y.G. and C.Y.; visualization, S.Z. and T.L. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by the National Science Fund Project (China) ’Unsteady flow and its excitation mechanism in stator and rotor cascades of nuclear main pump’ (51866009).

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: Not applicable.

Acknowledgments: We thank Jiangsu YaTai pump and valve Co., Ltd. for its strong support of our experiment.

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

Nomenclature

<table>
<thead>
<tr>
<th>Symbols of axial flow pump parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Symbol</strong></td>
</tr>
<tr>
<td>(A)</td>
</tr>
<tr>
<td>(L)</td>
</tr>
<tr>
<td>(c)</td>
</tr>
<tr>
<td>(g)</td>
</tr>
<tr>
<td>(C)</td>
</tr>
<tr>
<td>(H)</td>
</tr>
<tr>
<td>(n)</td>
</tr>
<tr>
<td>(p)</td>
</tr>
<tr>
<td>(Q_v)</td>
</tr>
<tr>
<td>(R)</td>
</tr>
</tbody>
</table>
Diameter, m
Radial coefficient
Pitch, m
Circumferential velocity, m/s
Absolute Velocity, m/s
Relative velocity, m/s
Blade number
Absolute flow angle
Relative flow angle
Hub ratio
Setting angle
Chord angle
Thickness, m
Crowding coefficient
Cascade density
Efficiency
Velocity coefficient
Flow coefficient
Head coefficient

\( R^* \)
\( t \)
\( U \)
\( V \)
\( W \)
\( Z \)
\( a \)
\( \beta \)
\( \epsilon \)
\( \kappa \)
\( \gamma \)
\( \delta \)
\( \zeta \)
\( \sigma \)
\( \eta \)
\( \varphi \)
\( \gamma \)
\( \psi \)

\( (R - R_h)/(R_i - R_h) \)
\( 2\pi R/Z \)
\( \tan^{-1}(\varphi_{in}/(R/R_i)) \)
\( \tan^{-1}(\varphi_{out}/(R/R_i)) \)
\( R_h/R_i \)

Cascade density/\( \sigma \)
Velocity coefficient/\( V/U_i \)
Flow coefficient/\( \gamma \)
Head coefficient/\( \psi \)

Name/Symbol
Radial coefficient/R*
Pitch/t
Inlet relative fluid angle/\( \beta_1 \)
Inlet absolute fluid angle/\( \beta_1 \)
Hub ratio/\( \epsilon \)

Equation

Name/Symbol
Equation

References


