Analysis of Internal Flow Characteristics of Water-jet Propulsion under Turning Conditions

Jiahao Chen1, Zhongyong Pan† and Xu Wu1

1 The Research Center of Fluid Mechanical Engineering Technology, Jiangsu University, Zhenjiang, 212013, China

†Corresponding author e-mail: pzy@ujs.edu.cn

Abstract: An axial water-jet propulsion was studies to explore the internal flow characteristics under turning conditions. Based on software ANSYS CFX, using the Ω vortex identification method, numerical simulation was carried out on the water-jet propulsion under different turning conditions for a fixed ship speed of 30 knot and the influence of different turning conditions on the internal flow field of the water-jet was obtained. In addition, monitoring points are set up at the impeller inlet and outlet respectively, and the internal pressure pulsation law of the water-jet propulsion is analyzed under different turning conditions. The calculation results show that the propulsion performance of the water-jet propulsion decreases with the increase of the turning angle, and it is that the performance of left-turning is better than that of right-turning. The vortex in the suction duct of the water-jet propulsion is mainly distributed around the rotating shaft, the inlet of the suction duct and the side wall opposite to the turning direction. The vorticity area of the impeller section increases with the increase of the turning angle, the suction surface vortex decreases with the increase of the turning angle in the right turning condition, and the trailing edge vortex increases with the increase of the turning angle. The operation condition of left turn is opposite to that of right turn. Affected by the turning angle, the pressure change at the impeller inlet flow surface is less severe than that of right turning. At the outlet surface of the impeller, the pressure pulsation regularity of the left turn condition is slightly better than that of the right turn, indicating that the flow at the outlet flow surface of the impeller under the left turn condition is better than that of the right turn.

1. Introduction
The water-jet propulsion system is a novel method of propelling ships forward by utilizing the reactive force of water flow. During a ship's maneuvering and turning, the inlet flow path of the water-jet propulsion system can become deflected, resulting in non-uniform flow patterns and altering the internal flow characteristics of the system. This, in turn, can impact the propulsion efficiency and operational
stability of the water-jet propulsion system. Therefore, investigating the internal flow of the water-jet propulsion system under different turning conditions is of significant importance[1].

Zhang et al conducted numerical simulations to investigate the trajectory of leakage vortices at the blade tip of axial flow pumps. The results indicated that the shear layer and jet at the blade tip were significantly influenced by the pressure difference between the pressure side and suction side, while the distribution of static pressure, turbulent kinetic energy, and vorticity within the TLV core was found to be associated with the TLV structure[2]. Cao et al utilized the Q-criterion vortex identification method to numerically analyze the water-jet propulsion system under non-uniform inflow conditions. The results revealed that large positive incidence angles near the blade inlet were the main cause of local separation, leading to a significant decrease in the water-jet pump's head[3]. Huang et al based on boundary vorticity analysis, investigated the characteristics of transient vortex cavitation in water-jet pumps under non-uniform inflow conditions and identified cavitation as an important mechanism for boundary vortex diffusion[4]. Bai et al examined the spatial evolution of leakage vortices at the blade tip of a straight hydrofoil using vorticity, the Q-criterion, Ω method, and Liutex method. They compared and explained the performance of different methods[5]. Zhao et al conducted numerical simulations to study the effects of rotational stall on the stability and pressure pulsation of centrifugal pumps. Through spectral analysis, they investigated the frequency and amplitude of pressure pulsations related to rotational stall, discovering that stall cells rotate circumferentially within the impeller and the interaction between stall cells and the splitter blade may influence Rotating Stall Induced (RSI) phenomena[6]. Cui et al elucidated the intrinsic correlation between unsteady flow structures and pressure pulsations in a low specific speed centrifugal pump by introducing a vortex identification method. The results indicated that the pressure pulsations at the impeller outlet were closely associated with the periodic shedding of vortices on the blade pressure side[7]. Luo et al investigated the mechanism of pressure pulsation induced by non-uniform inflow in water-jet pumps through numerical simulations. The results showed that due to the non-uniform inflow, the pressures near the inflow passage, impeller, and guide vane inlet fluctuated significantly at the impeller rotation frequency[8]. Jiao et al conducted experimental studies on the evolution of underwater suction vortices and pressure pulsation characteristics in water-jet propulsion systems using high-speed photography and pressure monitoring. The results revealed that underwater suction vortices could restore the amplitude of pressure pulsations after sudden drops, and the low-frequency pressure fluctuations induced by underwater suction vortices dominated the overall pulsation behavior[9]. Guo et al employed numerical simulations to reveal the characteristics of blade tip leakage flow and TLV cavitation under different cavitation conditions in water-jet pumps. They discovered that TLV and its cavitation reduced the blade tip leakage flow, and the analysis of pressure pulsations indicated that TLV cavitation enhanced the pressure pulsations[10].

Different navigation directions can cause changes in the direction of wave inflow, which in turn exacerbates the energy loss and reduces the performance of water-jet propulsion. It also leads to an increase in the unevenness of the internal flow field, an increase in pressure gradients, and an increase in the number of internal vortices. To investigate these issues, this paper employs numerical calculations and the Ω-vortex identification method to analyze the internal vortex distribution of water-jet propulsion under different turning conditions. Furthermore, pressure monitoring points are set at the inlet and outlet of the impeller to compare and analyze the effects of different turning angles on the internal flow of water-jet propulsion at a navigation speed of 30 knots.
2. Numerical calculation method

2.1 Computational Model and Mesh Irrelevance Verification

The specific parameters of the computational model in this paper are shown in Table 1:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water-jet pump</td>
<td></td>
</tr>
<tr>
<td>Blades</td>
<td>4</td>
</tr>
<tr>
<td>Guide vanes</td>
<td>7</td>
</tr>
<tr>
<td>Impeller diameter</td>
<td>285mm</td>
</tr>
<tr>
<td>Blade tip clearance</td>
<td>0.1mm</td>
</tr>
<tr>
<td>Suction duct</td>
<td></td>
</tr>
<tr>
<td>Wheel ratio</td>
<td>0.53</td>
</tr>
<tr>
<td>Outlet diameter</td>
<td>145mm</td>
</tr>
<tr>
<td>Width of water inlet</td>
<td>287mm</td>
</tr>
<tr>
<td>Height</td>
<td>328mm</td>
</tr>
</tbody>
</table>

The water-jet propulsion model was modelled using the 3D modelling software Creo. The model is shown in Figure 1.

![Figure 1. Water-jet propulsion model](image1)

To accurately simulate the operation of the water-jet propulsion system, a flow control body was added below the inlet duct to simulate the surrounding area of the bottom of the water-jet propulsion system. The dimensions of the flow control body are 30 times the diameter of the impeller in length, 10 times in width, and 8 times in height. The overall computational domain is shown in Figure 2.

![Figure 2. Water-jet propulsion calculation domain](image2)

The research model was divided using structured grids, and local grid refinement was applied to the blade region wall and clearance area. The suction duct was divided using unstructured grids, and it was ensured that the mesh numbers at the intersection of the grids were at the same level. This approach can achieve the best mesh quality and minimum angle adjustment, as well as ensure that the wall y+ meets the requirements for turbulence model calculations. ICEM was used to divide the flow control body into structured grids, as shown in Figure 3.

![Figure 3. Grid of inlet channels and water-jet pumps](image3)

Using the same mesh topology and numerical settings, the number of grids in the computational domain of the water-jet propulsion system is controlled by adjusting the mesh size and the number of nodes on the corresponding topological lines. Table 2 shows the grid independence test and the
performance parameters of the water-jet propulsion system for different numbers of grids. When the number of grids in the computational domain is 5,866,241, the change in the thrust of the water-jet propulsion system is less than 1%, which meets the requirements for grid independence. Finally, mesh scheme 4 was selected, with a first-layer mesh height of 0.01mm and a maximum y+ value of 48. The distribution of y+ in the impeller section is shown in Figure 4.

<table>
<thead>
<tr>
<th>Mesh division scheme</th>
<th>Number of grids</th>
<th>Thrust (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3164870</td>
<td>10783</td>
</tr>
<tr>
<td>2</td>
<td>3896547</td>
<td>12152</td>
</tr>
<tr>
<td>3</td>
<td>4643125</td>
<td>13269</td>
</tr>
<tr>
<td>4</td>
<td>5866241</td>
<td>14744</td>
</tr>
<tr>
<td>5</td>
<td>6304752</td>
<td>14690</td>
</tr>
<tr>
<td>6</td>
<td>6616417</td>
<td>14678</td>
</tr>
</tbody>
</table>

**Table 2. Grid irrelevance test**

2.2 Introduction to the Ω-vorticity identification method
Based on Liu et al.’s viewpoint, the vorticity in the flow field can be decomposed into rotational vorticity and non-rotational vorticity, and a new variable parameter Ω is introduced to represent the proportion of rotational vorticity[11]. The calculation formula of Ω is shown in Equation (1):

\[
\Omega = \frac{b}{a + b + \varepsilon}
\]

where \(a = \text{trace}(A^T A)\), \(b = \text{trace}(B^T B)\), \(A\) represents the symmetric tensor in the velocity gradient tensor, \(B\) represents the anti-symmetric tensor, and \(a\) and \(b\) respectively represent the square of their Frobenius norm. \(\varepsilon\) is a small positive number to prevent the denominator from being zero. In this paper, it is set to \(1 \times 10^{-3} \text{ s}^2\). It can be seen from the equation that \(1 > \Omega > 0\), and when \(\Omega > 0.5\), it is considered that the rotational vorticity dominates the flow field, indicating the existence of a vortex structure. An empirical reference value of \(\Omega = 0.52\) is given. This method achieves the dimensionless of the parameter, and obtains both strong and weak vortex structures, and solves the problem of artificially determining a suitable threshold in traditional vortex identification methods such as the Q-criterion and the \(\lambda_2\)-criterion[12].

2.3 Monitoring points and boundary condition settings
The numerical simulations in this study were conducted using the CFX software, with the SST k-ω turbulence model selected. In the computational domain, the impeller speed was set to 3300 r/min, and the flow control body, inlet channel, guide vanes, and nozzle were all set as stationary domains, while the walls of the computational domain were set as no-slip walls. The outlet of the flow control body was set as a free outflow boundary, in contact with the ocean waves, while the nozzle outlet was set as a pressure outlet with a pressure value of 0.1 MPa. The inlet of the control body was set as a velocity inlet.

Three monitoring points were set up at the impeller inlet and outlet of the water-jet propulsion, distributed at the hub, the middle of the blade inlet and the tip of the blade respectively. They were named M1, M2, M3, M4, M5 and M6 and their locations are shown in Figure 5. Based on the steady state numerical calculations, the interface between the rotating and stationary parts was changed to a "Transient rotor stator" and 10 impeller rotation cycles were simulated. The impeller was calculated every 5 degrees, corresponding to a time step of ΔT = 0.000252525s, for a total of 720 steps. The ninth cycle was chosen for analysis in order to study the pressure fluctuations at some of the impeller monitoring points relative to the angle of rotation.

![Figure 5](image)

**Figure 5.** Distribution of pressure pulsation monitoring points

### 2.4 Thrust performance tests

To ensure the accuracy of the numerical simulation, the thrust performance of the water-jet propulsion was tested through a static water test bench, where the diesel engine was used to change the rotational speed of the impeller and test the thrust performance at different speeds. Due to limited engine power, the impeller speed during the experiment was only able to reach the rated speed, without exceeding it. Multiple data collections were performed, and as shown in Figure 6, the thrust of the water-jet propeller increased with the increase of rotational speed, but the simulated values were consistently greater than the experimental values. The difference between the simulated and experimental values decreased as the speed approached the rated speed, with only 2% error at the rated speed of 3300r/min, indicating the reliability of the numerical simulation method proposed in this paper.
3. Calculation results and analysis

3.1 Comparison of Performance Curves at Different Turning Angles

Taking straight navigation as an example, the angle between the inflow direction and the flow direction of the water-jet propulsion is 0°. In order to investigate the influence of different turning directions and angles on the internal flow of the water-jet propulsion, this study selected the maximum turning angles of 30° to the left and right for investigation, and selected the working conditions of 0°, ±10°, ±20°, and ±30° for comparison.

As shown in Figure 7, the performance of the water-jet propulsion varies significantly under different turning angles. At a speed of 30 knots, the water jet propulsion performs optimally when traveling straight ahead (0° turning angle), while the internal flow deteriorates gradually as the turning angle increases. This leads to a reduction in flow rate, head, efficiency, and thrust output of the water jet pump. Additionally, under the same turning angle conditions, the water-jet propulsion performs better during left turns compared to right turns. In fact, the thrust output of the water-jet pump during a left turn of 30° can even be higher than during a right turn of 10°. This is because the designed flow rate of the water-jet pump is approximately 2900m³/h, while the flow rate of the water-jet propulsion under turning conditions can only reach around 0.8Q. This means that the water jet pump operates under conditions that are close to stall in high-angle turning situations.
3.2 Analysis of the water-jet propeller vortex structure under turning conditions

During turning, different turning angles bring different radial velocities to the inflow of the suction duct. This radial velocity causes an increase in flow non-uniformity within the suction duct and changes the distribution of vortices and vortex cores inside the suction duct. As shown in Figure 8, the distribution of vortices inside the suction duct under turning conditions is relatively uneven. During a left turn of 10°, the vortices mainly distribute near the upper wall surface and the axis above the inlet of the suction duct. However, the velocity distribution of the vortices is relatively uniform and has a small impact on the internal flow field. On the other hand, during a right turn of 10°, the vortex quantity inside the suction duct significantly increases compared to that during a left turn of 10°. A small-scale vortex appears near the inlet of the impeller under the suction duct, and a low-speed vortex surrounding the axis appears near the wall surface above the shaft, which severely disrupts the stability of the flow field above the shaft. As the turning angle increases, the size of the vortex further expands. This is manifested by the low-speed vortex near the wall surface above the shaft gradually spreading to the mid-section of the suction duct and the upper wall surface of the water passage, while the vortex near the impeller inlet under the suction duct gradually shifts upward and spreads toward the shaft area.

The spatial distribution of vortices under different turning angles shows certain regularities. Low-speed vortices are uniformly distributed in a surrounding manner near the wall in the front half of the
shaft. For vortices under the suction duct close to the impeller inlet, under right-turning conditions, they are mainly distributed near the lower-right wall of the impeller inlet; under left-turning conditions, they are mainly distributed near the lower-left wall of the impeller inlet. The velocity of the vortices also shows certain regularities, the velocity of the vortices decreases from the inlet of the suction duct upwards, and the velocity of the vortices above the shaft is lower than that below the shaft. The low-speed vortex situation is partially improved near the impeller inlet. Under left-turning conditions, the vortex velocity at the inlet of the suction duct is higher than that under right-turning conditions and is improved at the impeller inlet after the suction duct is rectified.

![Velocity distribution](image)

**Figure 8.** Vortex structure distribution in the inlet flow passage under turning conditions

To further investigate the effect of turning angle on the flow characteristics inside the suction duct, seven sections (L1–L7) were taken along the suction duct from the inlet to the impeller inlet surface, as shown in Figure 9. L1 corresponds to the inlet flow surface while L7 corresponds to the impeller inlet surface.
Figure 9. Distribution of monitoring sections along the inlet flow passage

Figure 10 illustrates the velocity streamline distribution at different cross-sections of the suction duct. It can be observed that the rotating vortex below the shaft exhibits an asymmetric distribution, mainly located near the upstream wall. In the right turning condition, the vortex is close to the right wall of the suction duct, while in the left turning condition, it is close to the left wall. The fluid velocity continuously increases in the flow direction, while the vortex size decreases. The counterclockwise vortex is eventually affected by the clockwise rotation of the impeller, leading to the disappearance of the reverse vortex. The flow state above the shaft is more severe than that below it. The vortex first appears at the front end of the shaft symmetrically towards the flow above it. At the middle section of the shaft, the vortex reaches its maximum size. As the flow approaches the impeller inlet, the vortex position gradually shifts along the direction of impeller rotation, and the vortex size decreases while the fluid velocity increases.

The internal pressure gradient within the suction duct increases with increasing turning angle, and the position of the internal vortex gradually shifts towards the low-pressure region. Under the left turning condition of 30°, four low-speed vortex clusters of similar size and close proximity appear above the shaft, resulting in a decrease in the outflow mass of the suction duct. This is primarily because the inflow velocity vector under the turning condition differs from that under the straight flight condition in only the axial velocity. The inflow velocity vector can be divided into axial velocity and radial velocity that is in the same direction as the turning direction. A portion of the radial velocity strikes the wall and is countered by the wall, causing local high pressure at the inlet of the flow passage. Another portion of the radial velocity is merged into the flow. Under the left turning condition, the fluid’s radial velocity is in the same direction as the impeller rotation direction, which brings about partial positive excitation pre-swirl at the impeller inflow face. However, under the right turning condition, the fluid’s radial velocity is opposite to the impeller rotation direction, and the impeller must do work to counteract this part of the radial velocity, resulting in energy loss.
Figure 10. Velocity streamline distribution at various cross-sections of the inlet flow passage

In the turning condition, the vortices on the blades of the water-jet propulsion are mainly distributed on the suction side, as shown in Figure 11, where the distribution of vortices on the suction side of the blade is shown. The vortices on the suction side of the blade are mainly distributed at the blade inlet edge and near the blade tip along the suction side. Inside the suction duct, vortices exist in both the upper and lower sections of the rotation axis. After the water flows into the impeller section, the vortices in the lower section of the rotation axis gradually disappear, and only a portion of high-speed vortices appear near the inlet edge of the suction side of the blade in the lower section of the rotation axis. The high-speed vortices are located at the blade inlet edge and their velocity decreases in the direction of water flow. In the right turn condition (positive turning angle), the area of the vortex near the blade inlet edge on the suction side of some blades decreases as the turning angle increases. However, in the left turn condition (negative turning angle), the area of the vortex on the suction side of the blade in the upper section of the rotation axis and on the right side of the rotation axis remains almost constant as the turning angle changes, while only the area of the vortex near the blade tip on the suction side of the blade on the left side of the rotation axis increases as the turning angle increases. Although the spatial distribution of vortices on the suction side of the blade in the left and right turn conditions of the water-jet propulsion is similar, the area and velocity of the vortices change almost inversely with the turning angle, and under the same turning angle condition, the area and velocity of the vortices on the suction side of the blade in the left turn condition are larger than those in the right turn condition.
Figure 11. Impeller suction surface vortex distribution

Figure 12 shows the distribution of vortices in the impeller section at different turning angles. The vortices mainly distribute near the impeller inlet hub, the suction side of the blade, and the near-tip position of the blade trailing edge. The velocity and structure of the vortices at the impeller hub do not show significant changes with the turning angle, while the distribution of vortices near the trailing edge varies noticeably with the turning angle.

Under the condition of a 10° right turn, the trailing vortices only exist in the flow passage I. As the turning angle increases to 20°, the distribution and area of the trailing vortices in flow passage I remain unchanged, but evident trailing vortices appear in flow passage II with a similar distribution and area to those in flow passage I. These trailing vortices all appear in the region near the trailing edge and the pressure side of the blade tip. When the turning angle increases to 30°, the scale of the trailing vortices in both flow passages I and II increases significantly, and they merge near the blade tip at the outlet of the impeller, forming a large-scale outflow low-speed vortex. Under the condition of a 10° left turn, the distribution of trailing vortices is similar to that of the right turn, but the area of the trailing vortices is slightly smaller than that of the right turn. The area of the trailing vortices in flow passage I decreases as the turning angle increases, and the vortex velocity also decreases to some extent. Additionally, no evident trailing vortices appear in flow passage II. The spatial distribution of the trailing vortices in the left and right turn conditions of the water-jet propulsion system is similar, but the area of the vortices varies almost inversely with the turning angle. The area of the trailing vortices in the right turn condition increases with the increase of turning angle, while in the left turn condition, the area decreases with the turning angle.
As shown in Figure 13, the relative area of the vorticity region in the blade passage increases with the turning angle, but the variation is relatively small, with a maximum increase of only 6.7% compared to the 0° straight navigation condition. Furthermore, at the same turning angle, the relative area of the vorticity region in the left-turn condition is slightly larger than that in the right-turn condition. Numerically, the relative change of the vorticity region inside the impeller is small. Therefore, in a sense, it can be inferred that as the turning angle increases, the vorticity in the impeller section under the right-turn condition transfers from the suction surface of the blade to the trailing edge, while under the left-turn condition, the vorticity in the impeller section transfers from the trailing edge to the suction surface of the blade.

\[ C_v = \frac{p - p_{avg}}{0.5\rho u_i^2} \]  

Where \( p \) the instantaneous pressure at the monitoring point, in Pa; \( p_{avg} \) the average pressure at the

---

**Figure 12.** Distribution of blade vortices

**Figure 13.** Relative area of the impeller section vortex area

### 3.3 Analysis of water jet propulsion pressure pulsation in turning conditions

The performance of the water-jet propulsion varies considerably under different turning angles, so the pressure pulsation of the water-jet propulsion is analysed under different turning angles and directional operating conditions. The monitoring pressure is dimensionless to obtain the pressure pulsation coefficient \( C_p \) the specific expression of the instrument is as follows:

\[ C_p = \frac{p - p_{avg}}{0.5\rho u_i^2} \]  

Where \( p \) the instantaneous pressure at the monitoring point, in Pa; \( p_{avg} \) the average pressure at the
monitoring point in time, in Pa; $\rho$ the density of the medium, in kg/m$^3$; $u^2$ the circumferential velocity at the impeller outlet, in m/s.

3.3.1 Time-domain analysis of pressure pulsation at the inlet and outlet of the impeller under turning conditions.

Figure 14 presents the time-domain plots of pressure pulsation at monitoring points M1, M2, and M3 on the intake surface of the impeller during one revolution under different turning directions and angles. The plots reveal a regular pattern of four peaks and valleys for pressure pulsation at each monitoring point throughout one cycle, with the number of cycles corresponding to the number of impeller blades. However, there is a noticeable variation in the magnitude of pressure pulsation within one cycle. Under right-turning conditions, the differences in pressure pulsation amplitudes among different turning angles are relatively small. Moreover, the closer the monitoring point is to the top region of the impeller, the smaller the differences in pressure pulsation amplitudes among different turning angles. Under left-turning conditions, there is a significant variation in pressure pulsation amplitudes. The positions of the peaks alternate between -10° and -20°, and the closer the monitoring point is to the top region of the impeller, the more stable the pattern of pressure pulsation curves. Among the monitoring points on the intake surface of the impeller, M1 exhibits the highest pressure pulsation amplitude, followed by M3, while M2 shows the lowest amplitude. This indicates that the pressure pulsation are greatest at the impeller hub near the intake, and the trend of pressure pulsation along the hub-to-tip direction shows a decrease followed by an increase. These findings suggest that, at the impeller inlet surface, the left-turning condition induces less severe pressure variations compared to the right-turning condition. The left-turning condition primarily affects the amplitude of pressure pulsation, while the right-turning condition has a relatively smaller impact on pressure pulsation. Furthermore, pressure pulsation diminish as one moves closer to the impeller tip, indicating a more stable inflow condition.

At monitoring point M1, under the same turning angle conditions, the right-turning condition consistently exhibits higher pressure pulsation amplitudes than the left-turning condition. However, within one cycle, the pressure pulsation amplitude shows significant variation for the left-turning condition at 10°, while the amplitudes for the 20° and 30° left-turning conditions remain relatively stable. In the first half of the cycle, the left-turning condition at 10° shows the largest pressure pulsation amplitude, followed by 30°, and the amplitude is smallest at 20°. In the second half of the cycle, the pressure pulsation amplitude for the left-turning condition at 10° sharply decreases to a minimum. At monitoring point M2, the pressure pulsation amplitudes for the left-turning and right-turning conditions are approximately equal in magnitude, and both exhibit the highest amplitudes at a turning angle of 20°. For the left-turning condition, the pressure pulsation amplitude is minimal at -10°, while for the right-turning condition, it is minimal at 30°. At monitoring point M3, significant differences are observed between the left-turning and right-turning conditions, with the left-turning condition showing higher pressure pulsation amplitudes than the right-turning condition. The pressure pulsation amplitudes for the right-turning condition exhibit minimal variation with changing turning angles. The pressure pulsation curve at M3 is smoother compared to M1 and M2, and within one cycle, it consistently shows a decrease in amplitude with increasing turning angles.
Figure 14. Time-domain distribution of pressure pulsation at the inlet surface of the impeller under different turning angles

Figure 15 depicts the time-domain plots of pressure pulsation at monitoring points on the exit surface of the impeller during turning conditions of various directions and angles. The graph reveals that the regularity of pressure pulsation decreases noticeably at the impeller outlet compared to the inlet. Under turning conditions, the pressure pulsation pattern is slightly better for left turns than for right turns. In some instances, certain monitoring points under right-turn conditions exhibit temporal deviations in pressure pulsation, with the pulsation trends appearing opposite for different turning angles. Moreover, the closer the points are to the hub, the more pronounced the temporal deviations in pressure pulsation become. However, the pressure pulsation amplitudes for left-turn conditions are consistently higher than those for right-turn conditions at all monitoring points. At the impeller outlet, the pressure pulsation amplitudes decrease initially and then increase along the hub-to-blade direction for left-turn conditions, whereas they continuously decrease for right-turn conditions. This indicates that the flow at the impeller
outlet surface is more favorable for left turns, and the amplification of pressure pulsation predominantly occurs at the end of a cycle. This behavior is primarily attributed to the interaction between the impeller and the stator vanes. The reduced regularity of pressure pulsation observed on the exit surface of the impeller during right turns may be primarily attributed to the enlargement of trailing edge vortices at the impeller outlet, which disrupts the internal flow conditions and obstructs the impeller exit, leading to a diminished regularity of pressure pulsation.

At monitoring point M4, during the first half of the cycle, the pressure pulsation amplitude is lowest for the left turn of 30°, with minimal numerical variations. The pressure pulsation amplitudes for left turns of 10° and 20° continue to increase. During the second half of the cycle, the pressure pulsation amplitude for the left turn of 30° shows a significant increase. Similarly, at monitoring points M5 and M6, the left turn of 30° exhibits a similar trend. It shows that the flow of the impeller outlet surface is relatively stable in the first half cycle. At monitoring point M5, there is no apparent pattern for the right turn condition, while the pressure pulsation amplitudes for the left turn condition exhibit smaller variations compared to points M4 and M6. However, there are slight temporal deviations in the pressure pulsation amplitudes. At monitoring point M6, there is also no apparent pattern for the right turn condition. For the left turn condition, the pressure pulsation amplitudes show a noticeable gradient, indicating that as the turning angle increases, the pressure pulsation amplitude also increases. Furthermore, the distribution of pressure pulsation amplitudes in the time domain generally aligns with this observation.

(a) Turn left M4

(b) Turn right M4

(c) Turn left M5

(d) Turn right M5
3.3.2 Frequency domain analysis of pressure pulsations at the impeller inlet and outlet under turning conditions

Figure 16 shows the frequency domain distribution of the pressure pulsation at the impeller inlet surface under different turning angles, and it can be found that the primary frequency at each monitoring point is located at the shaft frequency and the secondary frequency at the leaf frequency under different turning angles. The axial frequency and leaf frequency decrease from the hub to the top of the impeller, and the secondary peak phenomenon is obvious at the hub. This means that at the inlet, the pressure fluctuation at the hub decreases from the hub to the top of the leaf, which means that the flow at the hub is relatively unstable.

In the 10° right turn condition, the primary and secondary frequencies at monitoring point M1 are almost 50% higher than those at M2 and M3. As the turn angle increases to 30°, the main and sub frequencies increase, but the difference between the main and sub frequencies decreases. In the left turn condition, the trend of the main and sub frequencies at each monitoring point is similar to that of the right turn condition, but the change in values is more obvious. It can be found that when the turning angle is small, the main frequency of each monitoring point in the right turning condition is larger than that in the left turning condition with a 10° turning angle, and the sub frequency is smaller than that in the left turning condition. The pressure at the impeller inlet is more affected by the increased turning angle in the left-hand turning condition.
Figure 16. Frequency domain distribution of impeller inlet surface pressure pulsation at different turning angles

Figure 17 displays the frequency domain distribution of pressure pulsation at the blade inlet under different turning angles. It is evident that there are notable differences in the distribution of main and sub frequencies at the blade outlet between left and right turn conditions. Under the right turn condition, higher amplitude are observed at the blade frequencies and seven times the rotational frequency of the shaft. On the other hand, under the left turn condition, the main frequency is located at the rotational frequency of the shaft, while the sub frequency is located at the blade frequency. This indicates that the pressure pulsation at the blade outlet are primarily influenced by the dynamic interaction between the high-speed rotating blades and the stationary vanes in the right turn condition, whereas in the left turn condition, the pressure at the blade outlet is mainly influenced by the driving shaft.

Under the right turn condition at 10°, the amplitude difference between the blade frequency and seven times the rotational frequency of the shaft is relatively small at each monitoring point. However, the amplitudes at the blade frequency and seven times the rotational frequency of the shaft gradually increase from the hub to the blade tip. As the turning angle increases, the amplitudes at the blade
frequency and seven times the rotational frequency of the shaft decrease at each monitoring point. The amplitude at the blade frequency decreases from the hub to the blade tip, while the amplitude at seven times the rotational frequency of the shaft initially decreases and then increases from the hub to the blade tip. At monitoring point M6, the amplitude at seven times the rotational frequency of the shaft surpasses that at the blade frequency. Under the left turn condition at 10°, the main frequency decreases from the hub to the blade tip, followed by an increase, while the sub frequency is distributed at the blade frequency and seven times the rotational frequency of the shaft. With increasing turning angle, a clear pattern emerges in the frequency domain distribution of pressure pulsation. The amplitude of the main frequency increases by nearly 200% and exhibits a distinct secondary peak phenomenon. At this point, the pressure pulsation at the blade outlet are also influenced by the dynamic interaction between the blades and vanes, resulting in prominent peaks at the blade frequency and seven times the rotational frequency of the shaft. The main frequency continuously increases from the hub to the blade tip. This suggests that under the right turn condition, as the turning angle increases, the influence of the dynamic interaction between the blades and vanes on pressure pulsation gradually weakens, while the influence of the rotational shaft becomes more pronounced. Under the left turn condition, the influence of the driving shaft on pressure pulsation increases with the turning angle, and the effect of dynamic interaction between the blades and vanes is evident.

(a) Turn right 10°

(b) Turn left 10°
4. Conclusion

This study conducts numerical simulations on a water-jet propulsion under different turning angles to analyze its performance, vortex structure distribution, and pressure fluctuation characteristics. The objective is to investigate the effects of different turning directions and angles on the water-jet propulsion. The following conclusions are drawn:

1. With increasing turning angle, the flow rate, head, efficiency and thrust of the water-jet propulsion system all exhibit a decreasing trend, with better performance observed during left turns compared to right turns.

2. The suction duct of the water-jet propulsion system experiences non-uniform distribution of vortices and velocities. Vortices mainly concentrate around the drive shaft, the inlet of the suction duct, and the side wall opposite to the turning direction, leading to the occurrence of asymmetric cavitation. The relative area of vortices in the blade section increases slightly with increasing turning angle, but the increment is limited. An increase in left turning angle results in a larger vortex area on the suction surface of the blades, while an increase in right turning angle leads to a larger vortex area near the trailing edge of the blades. The flow inside the water inlet section exhibits slightly better characteristics in left turns compared to right turns, with lower mass flow quality at the impeller inlet and higher mass flow quality at the impeller outlet for left turns.

3. The pressure pulsation at the impeller inlet exhibits a higher level of regularity, with more stable flow closer to the impeller top over one cycle. In the vicinity of the hub, the pressure pulsation is stronger in right turns compared to left turns, while at other positions, the pressure pulsation intensity is relatively consistent. Within a single cycle, the pressure pulsation amplitude is greater for left turns compared to right turns. At the impeller outlet, no apparent pattern is observed for the pressure pulsation in the monitoring points of right turns, while for left turns, the pressure pulsation amplitude shows a clear gradient, where larger turning angles correspond to higher pressure pulsation amplitudes. Moreover, the pressure pulsation amplitudes for different turning angles exhibit similar temporal distributions, with an increased amplitude occurring at the end of one cycle. The enlargement of the trailing edge vortex disrupts the flow on the impeller outlet surface, resulting in a more chaotic pressure pulsation pattern.
for right turns.

References


