Numerical Simulation of Hydraulic Characteristics in A Vortex Drop Shaft

Wenchuan Zhang®, Junxing Wang *, Chuangbing Zhou, Zongshi Dong and Zhao Zhou

State Key Laboratory of Water Resources and Hydropower Engineering Science, Wuhan University, Wuhan 430072, China; zhangwenchuan@whu.edu.cn (W.Z.); cbzhou@whu.edu.cn (C.Z.); dzs@whu.edu.cn (Z.D); zh_whu@sina.com (Z.Z.)
* Correspondence: jxwang@whu.edu.cn; Tel.: +86-137-0718-2138

Received: 18 July 2018; Accepted: 25 September 2018; Published: 28 September 2018

Abstract: A new type of vortex drop shaft without ventilation holes is proposed to resolve the problems associated with insufficient aeration, negative pressure (Unless otherwise specified, the pressure in this text is gauge pressure and time-averaged pressure) on the shaft wall and cavitation erosion. The height of the intake tunnel is adjusted to facilitate aeration and convert the water in the intake tunnel to a non-pressurized flow. The hydraulic characteristics, including the velocity (Unless otherwise specified, the velocity in this text is time-averaged velocity), pressure and aeration concentration, are investigated through model experiment and numerical simulation. The results revealed that the RNG k-ε turbulence model can effectively simulate the flow characteristics of the vortex drop shaft. By changing the inflow conditions, water flowed into the vertical shaft through the intake tunnel with a large amount of air to form a stable mixing cavity. Frictional shearing along the vertical shaft wall and the collisions of rotating water molecules caused the turbulence of the flow to increase; the aeration concentration was sufficient, and the energy dissipation effect was excellent. The cavitation number indicated that the possibility of cavitation erosion was small. The results of this study provide a reference for the analysis of similar spillways.

Keywords: vortex drop shaft; turbulence model; spillway aerators; energy dissipation; cavitation

1. Introduction

Compared with traditional energy dissipators, vortex drop shaft spillways can rapidly change the flow regime and form areas of turbulence or whirlpools to dissipate energy. These spillways provide various functions, such as flood discharge control, energy dissipation and drainage structure protection. Additionally, they transfer the energy dissipation task from outside to inside to avoid outlet atomization (in the traditional discharge energy dissipation process, water interacts with the air boundary to form an atomized flow [1]). However, within these spillways, the pressure near the vertical shaft wall gradually decreases because of gravity and wall friction, and generated negative pressure can easily cause cavitation erosion [2,3]. Therefore, considerable attention has been paid to negative pressure and cavitation erosion in the applications of vortex drop shaft spillways. To solve these problems, erosion reduction devices are often installed in vortex drop shaft spillways to ensure the stable operation of the discharge structures.

Many studies have investigated the characteristics of vortex drop shaft spillways, mainly focusing on the diameter of the vertical shaft and the depth of the dissipation well; they found that the energy dissipation effect could be improved by optimizing the vertical shaft shape [4]. Design schemes of vortex chambers and vertical shafts were proposed based on previous research on vortex drop shaft spillways, and the design criteria was summarized [5,6]. Based on an analysis of the influence of the body shape on the hydraulic parameters of an energy dissipation well, one study found that a
reasonable depth for an energy dissipation well should be 1.69 times the vertical shaft diameter [7]. Other studies focused on the tangential slot vortex intakes of vertical shafts for urban drainage, and they found that the flow in a tapering and downward-sloping vortex inlet channel was strongly dependent on the geometry of the inlet and vertical shaft and that the hydraulic instability was related to the discharge [8,9]. Del Giudice and Gisonni [10] found that an appropriate length of the lowered bottom could prevent free surface fluctuations at the intake structure, resulting in an appropriate vortex air core without choking problems for supercritical approach flows. In addition, the hydraulic characteristics of swirling flows and the energy dissipation rate were both related to the swirl number. The larger the swirl number was, the greater the energy dissipation. Furthermore, the tangential velocity ensured no negative pressure on the vertical shaft wall and enhanced the flow stability [11]. The radial distribution of the pressure in the swirl zone complied with the theoretical distribution of the pressure in combined eddies, and the pressure increased as the radius increased [12]. In previous studies, the velocity of a cavity swirl was assumed based on the theory of combined vortex and free vortex, and a corresponding pressure formula was obtained [13,14]. In another investigation, the need for aeration facilities was emphasized, and a method for calculating the aeration cavity length of annular aeration in a swirling vertical shaft was derived based on projectile theory [15]. Dong et al. [16] used the standard $k$-$\epsilon$ turbulence model to simulate a vortex drop shaft spillway and obtained the hydraulic parameters of spiral flow. Gao et al. [17] simulated the characteristics of turbulent flow through a vertical pipe inlet/outlet with a horizontal anti-vortex plate. However, compared with the standard $k$-$\epsilon$ turbulence model, the RNG $k$-$\epsilon$ turbulence model can more effectively process flows with high strain rates and streamline bending by correcting the turbulent viscosity and considering both rotation and swirling in the average flow regime. In addition, a previous study found that the water surface was more stable using this approach, and the simulation results agreed well with experimental data [18,19].

For traditional vortex drop shaft spillways, the water in the intake tunnel moves with pressure flow, and thus ventilation holes are set in the volute chamber or in front of the outlet tunnel [2,5,20]. Moreover, an annular aerator was set in the middle of the shaft to increase the aeration concentration and avoid erosion [15]. In practical engineering, aeration facilities often cause great difficulties during the process of construction. Therefore, to resolve the insufficient energy dissipation, cavitation of traditional spillways and construction difficulty, a vortex drop shaft with an increased-height intake tunnel and without ventilation holes is investigated in the present study. The pressure field, flow field and energy dissipation are simulated and analyzed by the renormalization group (RNG) $k$-$\epsilon$ turbulence model [21], and the results are compared with experimental data to better understand the distribution trends of the hydraulic parameters, the characteristics of the energy dissipation and the cavitation associated with a vortex drop shaft.

2. Physical Model

A Froude similitude with a geometric scale ratio of 1:25 was applied in the physical model. The model scale has a slight effect on the time-averaged hydraulic characteristics such as the time-averaged water depth, pressure and velocity, and the scale effects could be negligible, so they can provide useful references for practical engineering [22]. Small-scale models based upon the Froude similitude may underestimate the air transport in the fluid because the relative effects of the surface tension and viscosity are over-represented, especially when the scale is smaller than 1:30 [23–26]. In summary, a model experiment can simulate the time-averaged water depth, pressure and velocity well, and the aeration concentration is underestimated so that it is larger in the prototype. A higher aeration concentration is more favourable for preventing cavitation corrosion, so the experimental results can be used as a reliable reference for engineering design.

The vortex drop shaft investigated in this study consists of a tangential intake tunnel, a volute chamber, a gradient section, a vertical shaft, a dissipation well, and an outlet tunnel. The intake tunnel has an arched shape, with a slope and length of 1:7.5 and 0.68 m, respectively, and the section size
changed from 0.2 m × 0.24 m (width × height) to 0.176 m × 0.33 m (width × height). The diameter of the volute chamber is 0.352 m, and the height from the gradient section to the top of volute chamber is 0.72 m. The diameter and depth of the vertical shaft are 0.216 m and 1.82 m, respectively. The depth of the dissipation well is 0.28 m and the height of gradient section is 0.24 m. The outlet tunnel has an arched shape, with a width, height, length and slope of 0.2 m, 0.24 m, 6.1 m and 1:50, respectively. There are no ventilation holes in the volute chamber, gradient section, or vertical shaft, and air flows with the water from the intake tunnel into the vertical shaft. The sloped section and the straight plate are set at the inlet of the outlet tunnel to decrease fluctuations in the water and ensure that the flow regime is open-channel flow. The structure is shown in Figure 1. To reveal the characteristics of the energy dissipation and the cavitation within the vortex drop shaft, three tests at flood frequencies of 5%, 2% and 0.1% (expressed as P = 5%, 2% and 0.1% in the following) were carried out in this paper. A fluvigraph (the margins of error are plus or minus 0.18 mm) is used to control the upstream water level, and the tailwater depth in the outlet tunnel is controlled by a valve located in the downstream channel. Eighteen measurement points are established in the wall of the vertical shaft, as shown in Figure 1, the odd and even numbers located upstream and downstream of the vortex drop shaft, respectively; the measurement locations are close to wall.

In the present study, the experimental velocity was measured by a small specially designed L-shaped tube [27] similar to a Pitot tube, which is 160 mm long and 2 mm in diameter with a short inlet as shown in Figure 2. The margins of error are plus or minus 5 percent. The L-shaped tube was aligned with the direction of incoming flow by visual measurement, and the result was carried out after the liquid column height reached the maximum and was stable. Time-averaged pressure was measured by a liquid column manometer based on hydrostatic principle, and the accuracy is 0.1mm. The aeration concentration was measured with a CQ6-2004 aeration concentration meter (China Institute of Water Resources and Hydropower Research, Beijing, China), and a single-chip microcomputer was used to acquire and process the data. The aeration concentration is determined by detecting the resistance of water and aerated water between two electrodes. The resolution is 0.1% and the sampling frequency is 1020 Hz.
3. Numerical Simulation

3.1. Turbulence Model

Considering the advantage of the free interface capture and the specialized air entrainment model inside [28–30], Flow-3D (Flow Science, Santa Fe, NM, USA) was chosen for the numerical simulation. The RNG k-ε turbulence model and the volume of fluid (VOF) method [31] are used for the turbulence model and free surface tracking, respectively. Numerical discretization is performed with the finite difference method. The algebraic equations are solved using the generalized minimum residual (GMRES) method. The fluid is assumed to be incompressible, the governing equations are presented as follows:

Continuity equation

$$\frac{\partial u_i}{\partial x_i} = 0$$  \hspace{1cm} (1)

Momentum equation

$$f_i - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} \right) = \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j}$$  \hspace{1cm} (2)

$k$ equation

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k + \rho \epsilon$$  \hspace{1cm} (3)

$\epsilon$ equation

$$\frac{\partial (\rho \epsilon)}{\partial t} + \frac{\partial (\rho u_i \epsilon)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \alpha_\epsilon \mu_{eff} \frac{\partial \epsilon}{\partial x_j} \right) + \frac{C^*_\epsilon}{k} \frac{k}{\epsilon} G_k - C_{2\epsilon} \rho \frac{\epsilon^2}{k}$$  \hspace{1cm} (4)

The fluid configurations are defined in terms of a VOF function $F(x,y,z,t)$, which represents the VOF per unit volume and satisfies the following equation:

$$\frac{\partial F}{\partial t} + \frac{\partial}{\partial x_i} (F u_i) = 0$$  \hspace{1cm} (5)

where $u_i$ and $x_i$ are the time-averaged velocity and coordinate components, respectively. $f_i$ is the time-averaged component, $t$ is the time, $\mu$, $\nu$ and $\rho$ are coefficient of dynamic viscosity, kinematic viscosity and density, respectively. $G_k$ is the generation of turbulent energy caused by the average velocity gradient, $p$ is the gauge pressure, $\mu_{eff}$ is the revisionary coefficient of dynamic viscosity, $F$ is the VOF function, $\mu_{eff} = \mu + \mu_t$, $G_k = \mu_t \left( \frac{\partial u_i}{\partial x_i} + \frac{\partial u_j}{\partial x_j} \right) \frac{\partial u_i}{\partial x_j}$, $\mu_t = \rho C_\mu \frac{k^2}{\epsilon}$, $C^*_\epsilon = C_{1\epsilon} - \eta \left( \frac{1 - \beta}{1 + \beta} \right)$, $\eta = (2\epsilon_{ij} \times E_{ij})^{\frac{1}{2}} \epsilon$, $E_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$. There are some constants provided by Launder [32] and verified by a large number of experiments: $C_\mu = 0.0845$, $\alpha_k = \alpha_\epsilon = 1.39$, $C_{1\epsilon} = 1.42$, $C_{2\epsilon} = 1.68$, $\eta_0 = 4.377$ and $\beta = 0.012$.

3.2. Air Entrainment Model

The air entrainment model in FLOW-3D was first proposed by Hirt in 2003 [33,34]. This model assumes that air entrainment at the free surface is caused by an instability force $P_l$ produced by the turbulence of the free surface. When the level of turbulence exceeds the stability force $P_d$, which is associated with gravity and surface tension, air with a volume $\delta V$ will be entrained into the water. The governing equations are as follows:

$$L_T = \frac{C_{LU}^{0.75} k^{1.5}}{\epsilon}$$  \hspace{1cm} (6)

$$P_l = \rho k; \quad P_d = \rho g u L_T + \sigma \frac{\epsilon}{L_T}$$  \hspace{1cm} (7)
\[
\delta V = \begin{cases}
C_{air}A_s \left( \frac{z(P_t - P_d)}{\rho} \right) & \text{if } P_t > P_d \\
0 & \text{if } P_t < P_d
\end{cases}
\]  

where \( L_T \) denotes the turbulence length scale, \( C_{NU} \) is a constant equal to 0.09, \( k \) and \( \varepsilon \) are the turbulent kinetic energy and turbulent dissipation rate, respectively. \( g_n \) is the component of gravity normal to the free surface, \( \sigma \) is the coefficient of surface tension, \( \delta V \) is the volume of air entrained per unit time, \( C_{air} \) is a coefficient of proportionality and \( A_s \) represents the surface area.

3.3. Computational Grid and Boundary Conditions

For comparison with the experimental results, the upstream (Y-Min) and top (Z-Max) boundaries are set as pressure boundaries, and the water elevation at the upstream boundary is added, the downstream (Y-Max) boundary is set as the outflow. The left (X-Min), right (X-Max) and bottom (Z-Min) boundaries are set as solid, non-slip wall boundaries. Moreover, the boundaries of the nested grids are set as symmetry, as shown in Figure 3a. The model includes 2 mesh blocks, and the nested grids are used to ensure the accuracy of grid segmentation in the calculation, as shown in Figure 3b. To evaluate the grid independence of the results, the discharges tests were undertaken with different grids. Four schemes were carried out to verify grid accuracy in Table 1, the difference between grids 3 and 4 was 1.12%, which suggested that grid 4 was already sufficient for the simulation. The time step is variable, and the initial time step and residual error are \( 1 \times 10^{-7} \) and \( 1 \times 10^{-6} \), respectively. Additionally, the termination time is set as the time at which the rate of change in the total volume of the fluid is less than 0.1%, and simulation time is 200 s in this paper.
Table 1. The results of grid independence (P = 0.1%).

| Grid | Description | Size (X × Y × Z) | Discharge (L/s) |
|------|-------------|-----------------|-----------------|
| 1    | containing block | 15 mm × 15 mm × 15 mm | 54.47 |
|      | nested block    | 10 mm × 10 mm × 10 mm | |
| 2    | containing block | 10 mm × 10 mm × 10 mm | 58.25 |
|      | nested block    | 5 mm × 5 mm × 5 mm | |
| 3    | containing block | 5 mm × 5 mm × 5 mm | 60.28 |
|      | nested block    | 3 mm × 3 mm × 3 mm | |
| 4    | containing block | 4 mm × 4 mm × 4 mm | 60.96 |
|      | nested block    | 2 mm × 2 mm × 2 mm | |

4. Results and Discussion

4.1. Model Verification

Liu et al. [19] found that both the standard and RNG $k$-$\varepsilon$ turbulence models could simulate the flood movement of the vortex drop shaft spillway well and reproduced the water flow state of the swirling flow cavity in the vertical shaft; however, the RNG $k$-$\varepsilon$ turbulence model predicted pressure and flow velocity better than the standard $k$-$\varepsilon$ model. Figure 4 shows that the pressure and velocity distributions of the vortex drop shaft calculated using the two models have similar patterns. The pressure and velocity values calculated in the vertical shaft were close and in rather good agreement with the experiment; however, the RNG $k$-$\varepsilon$ turbulence model generated results closer to those obtained in the experiment, especially in the dissipation well. The relative deviations for pressure of the standard and RNG $k$-$\varepsilon$ turbulence models with respect to the model test results were 18% and 11%, respectively, and the relative deviations for velocity were 8% and 5%, respectively. In general, the RNG $k$-$\varepsilon$ turbulence model can better simulate the flow of the vortex drop shaft than the standard $k$-$\varepsilon$ turbulence model, indicating that the RNG $k$-$\varepsilon$ turbulence model is more suitable for dealing with a distorted flow or rotational flow with a high strain rate. Therefore, the following analysis is based on the RNG $k$-$\varepsilon$ model.

Figure 4. Cont.
was able to model the main characteristics of the vortex drop shaft. Which indicated the flow was draining freely into the drop shaft and was supercritical along the entire intake tunnel. Even though there were some errors between experiment and simulation due to the surface fluctuations after aeration and unavoidable instrument error, in general, the numerical model was similar to the experimental results, which suggested that the numerical simulation results were reliable. Figure 5 shows the Froude number (the intake tunnel has an arched shape and the flow was non-pressure flow; because the flow can be viewed as the open-channel flow in a rectangular channel, the Froude number is calculated by \[ F_r = \frac{V}{\sqrt{gh}} \], where \( h \) is water depth) in the intake tunnel. The Froude number was larger than 1 and increased with the discharge, which indicated the flow was draining freely into the drop shaft and was supercritical along the entire intake tunnel. Even though there were some errors between experiment and simulation due to the surface fluctuations after aeration and unavoidable instrument error, in general, the numerical model was able to model the main characteristics of the vortex drop shaft.

4.2. Flow Regime

The discharges, which depend on the flood elevation upstream, are obtained through via flow meter and baffle (the baffle is a tool in Flow-3D). Because of measurement error in flow meter and computational grid, small differences were unavoidable. Overall, the errors between experiment and simulation were less than 3% (the maximum error is 2.14% at the flood frequency of 0.1%) as shown in Table 2. The numerical results were similar to the experimental results, which suggested that the numerical simulation results were reliable. Figure 5 shows the Froude number (the intake tunnel has an arched shape and the flow was non-pressure flow; because the flow can be viewed as the open-channel flow in a rectangular channel, the Froude number is calculated by \( F_r = \frac{V}{\sqrt{gh}} \), where \( h \) is water depth) in the intake tunnel. The Froude number was larger than 1 and increased with the discharge, which indicated the flow was draining freely into the drop shaft and was supercritical along the entire intake tunnel. Even though there were some errors between experiment and simulation due to the surface fluctuations after aeration and unavoidable instrument error, in general, the numerical model was able to model the main characteristics of the vortex drop shaft.

Table 2. Comparison of discharge between experiment and simulation.

| Flood Frequency | Discharge (L/s) | Deviation |
|-----------------|----------------|-----------|
|                 | Experiment    | Simulation|          |
| 5%              | 30.45         | 29.92     | 1.74%    |
| 2%              | 38.21         | 38.01     | 0.52%    |
| 0.1%            | 62.29         | 60.96     | 2.14%    |

![Figure 4. Pressure and velocity calculated using different turbulence models (P = 0.1%).](image)

![Figure 5. Froude number profile in intake tunnel.](image)
In the experiment, the water entered the intake tunnel in a state of open-channel flow, and it provided a height of 5.0-cm for the air inlet under the flood frequency of 0.1% (as shown in Figure 6). The discharged water flowed into the vertical shaft with a large amount of air to form a stable mixing cavity through the intake tunnel. The water surface at the downstream wall of the vortex chamber was higher than that at the upstream wall, and the surface exhibited some fluctuations, as shown in Figures 6 and 7. The cavity throat (i.e., the narrowest cavity section) occurred in the gradient section because of the decreases in the section size and discharge area. As the axial velocity increased in the vertical shaft, the water layer became thinner. Then, water dropped into the dissipation well at the end of the vertical shaft and formed a water cushion, which exhibited a large-scale swirling motion and intense mixing. An aeration phenomenon was obvious, and the flow was full of milky white bubbles. Highly turbulent flow was observed in the sloped section, air bubbles gradually drifted upward and out from the surface in the outlet tunnel, and the flow was stable.

![Cavity in the drop shaft](image1.png)  ![Dissipation well](image2.png)

(a) Cavity in the drop shaft  (b) Dissipation well

**Figure 6.** Flow conditions in the experiment (P = 0.1%).

![Volume fraction of fluid in the numerical simulation](image3.png)

(a) Longitudinal profile  (b) Transverse profile

**Figure 7.** Volume fraction of fluid in the numerical simulation (P = 0.1%).

### 4.3. Velocity Distribution

The flow velocity is an important parameter when analyzing the energy dissipation effect and calculating the energy dissipation ratio. The velocity in the vortex drop shaft was mainly affected by gravity, friction and shear force, which first increased and then decreased, and the velocity peaked
at the connection with the sloped section ($z = 0.70$ m). The maximum velocities were 2.50 m/s, 2.68 m/s, 3.45 m/s (at flood frequencies of 5%, 2% and 0.1%, respectively) in the experiment and 2.56 m/s, 2.89 m/s, 3.49 m/s in the simulation, as shown in Figure 8. At the bottom of the vertical shaft, the upper and lower flows collided, resulting in a rapid decrease in the velocity, indicating that the energy dissipation effect was sufficient.

![Figure 8. Velocity in the vortex drop shaft.](image)

The velocity field (flood frequency of 0.1%) is shown in Figure 9. At the same elevation, the velocity closer to the wall was smaller due to friction along the vertical shaft wall. When the water fell from the wall into the dissipation well, the maximum velocity was approximately 5.0 m/s. Subsequently, the discharged flow collided with the water cushion in the well, and the velocity rapidly decreased to 1.0 m/s at the bottom of the well and approximately 2.0 m/s at the tunnel outlet. Thus, the energy was effectively dissipated.

The direction of the velocity varied within the vertical shaft, and the average velocity was calculated based on a theoretical analysis to quantify the cross-sectional kinetic energy distribution. The velocity of a swirling flow can be decomposed into a tangential velocity, an axial velocity and a radial velocity. The radial velocity is often ignored because it is smaller than the tangential and axial velocities (generally by more than two orders of magnitude). Additionally, it is often assumed that swirling flow is associated with a free eddy and meets the conditions for the conservation of angular momentum.
Two sections in the vertical shaft are used to establish the Bernoulli equation, as shown in Figure 10:

\[
\frac{V^2}{2g} + dZ = \frac{(V + dV)^2}{2g} + \lambda dS \frac{V^2}{2gR^2}
\]

(9)

where the hydraulic radius is \( R' = \frac{A}{\pi R} \) and the water trajectory is \( dS = \frac{dZ}{\cos \theta} \). \( R \) and \( Q \) are the radius of vertical shaft and the flow discharge, respectively; \( g \) is the constant of gravitational acceleration; \( \theta \) is the angle between the direction of the velocity and the vertical direction; and \( \lambda \) denotes the factor of friction loss. Additionally, \( V \) is the resultant velocity, and \( dV \) is considered negligible in this study. The hydraulic radius and the water trajectory are substituted into Equation (9) to obtain Equation (10):

\[
\int_{Z_0}^{Z_1} d\left( \frac{V^2}{2g} \right) = \left( 1 - \frac{\pi \lambda R V^3}{2Q^2} \right) dZ
\]

(10)

with \( G^{3/2} = \frac{\pi \lambda R}{2Q^2} \left( \frac{V^3}{2g} \right)^{3/2} \) and \( K = \left( \frac{\pi \lambda R}{2Q^2} \right)^{2/3} \), we have \( G = K \frac{V^2}{2g} \). This formula can be substituted into Equation (10), and the new equation can then be integrated as follows:

\[
K \int_0^Z dZ = \int_{G_0}^{G} \frac{dG}{1 - G^{3/2}} = \int_0^{G_0} \frac{dG}{1 - G^{3/2}} - \int_{G_0}^{G} \frac{dG}{1 - G^{3/2}}
\]

(11)

\[
H(G) = H(G_0) + KZ
\]

(12)

\[
H(G) = \frac{2}{3} \ln \frac{1}{1 - \sqrt{G}} + \frac{1}{3} \ln \left( G + \sqrt{G} + 1 \right) - \frac{2}{\sqrt{3}} \left( \arctan \frac{2\sqrt{G} + 1}{\sqrt{3}} - \arctan \frac{1}{\sqrt{3}} \right)
\]

(13)

where \( K \) is a coefficient, \( G \) is a function related to \( R, Q, \lambda \) and \( V \), \( H \) is a function related to \( G \), \( Z \) is the length of the shaft from the gradient section.

The initial velocity was obtained by measuring the velocity in the vertical shaft (at \( z = 0 \), as shown in Figure 10) several times. Then, the average velocity in each section was calculated by combining Equations (12) and (13). In the numerical simulation, the average velocity was measured by establishing a flux plane (baffle) in the measurement section. The average velocities obtained based on the theoretical calculations and simulations were compared in Figure 11. This figure shows that the analytically and numerically obtained resultant velocities were in good agreement. Both results also had similar patterns of variation and the mean error between the theoretical and simulated values were 3.81%, 3.63% and 5.65% (at flood frequencies of 5%, 2% and 0.1%, respectively). In general, there was a
high consistency between the theoretical and numerical results. The developed theoretical calculation method for flow velocity can function as a reference for vertical shaft design.

![Diagram of the average velocity in the shaft section.](image)

**Figure 10.** Diagram of the average velocity in the shaft section.

4.4. **Pressure Distribution**

Cavitation is a phenomenon that may occur when the local pressure extends below the vapor pressure of the liquid at the operating temperature. When cavitation occurs, collapsing cavitation bubbles can cause material erosion, thereby decreasing the lifespan and performance of the vortex drop shaft spillway [35]. Therefore, the pressure is one of the most important indicators of whether a vortex drop shaft can stably operate; consequently, negative pressure zones should be avoided or as small as possible. The time-averaged pressure along the vortex drop shaft exhibited an obvious distribution characteristic, being large at the top and bottom but small at the middle as shown in Figure 12. The minimum pressures were 0.04 kPa, 0.13 kPa, 0.23 kPa (at flood frequencies of 5%, 2% and 0.1%, respectively) in the experiment and 0.07 kPa, 0.12 kPa, 0.18 kPa in the simulation at \( z = 0.7 \text{ m} \). Additionally, the pressures in the gradient section and dissipation well were obviously higher than that in the vertical shaft. The wall pressure in the vertical shaft was caused by a centrifugal force, and during the falling process, the tangential velocity and centrifugal force gradually decreased, which subsequently led to the decreased pressure.

![Comparison of the simulated and theoretical average velocity in the shaft section.](image)

**Figure 11.** Comparison of the simulated and theoretical average velocity in the shaft section.
and 0.1%, respectively) in the experiment and 0.07 kPa, 0.12 kPa, 0.18 kPa in the simulation at z = 0.7 m. Additionally, the pressures in the gradient section and dissipation well were obviously higher than that in the vertical shaft. The wall pressure in the vertical shaft was caused by a centrifugal force, and during the falling process, the tangential velocity and centrifugal force gradually decreased, which subsequently led to the decreased pressure.

Figure 12. Pressure trends in the vertical shaft.

An area of negative pressure formed at the connection between the vertical shaft and outlet tunnel because the swirling flow moved away from the vertical shaft wall as shown in Figure 13 (flood frequency of 0.1%). Moreover, the pressure in the dissipation well was larger, approximately 10 kPa, due to the influences of the hydrostatic pressure and discharge. The maximum pressure in the entire vortex drop shaft was observed at the bottom of the dissipation well, with experimental and simulated values of 13.1 kPa and 12.6 kPa, respectively (flood frequency of 0.1%). The similarity between the experimental and simulated values further verified the accuracy of the numerical simulation.

Figure 13. Pressure profiles based on the numerical simulations (P = 0.1%).

4.5. Aeration Concentration

In the vortex drop shaft without ventilation holes, the water flowed into the vertical shaft with a large amount of air and created a stable mixing cavity through the intake tunnel. Due to the rotation of the flow and the existence of the cavity, air was continuously drawn into the cavity, and the velocity was high in the cavity area. In the experiment, the water layer was thin, and there was an obvious aeration phenomenon. The aerated water rushed into the dissipation well at a high speed and there was considerable velocity fluctuation in the water cushion. The aeration concentration was measured with a CQ6-2004 aeration concentration meter, and the results are shown in Figure 14.

Figure 14. Aeration concentration in the vortex drop shaft.
4.5. Aeration Concentration

In the vortex drop shaft without ventilation holes, the water flowed into the vertical shaft with a large amount of air and created a stable mixing cavity through the intake tunnel. Due to the rotation of the flow and the existence of the cavity, air was continuously drawn into the cavity, and the velocity was high in the cavity area. In the experiment, the water layer was thin, and there was an obvious aeration phenomenon. The aerated water rushed into the dissipation well at a high speed and there was considerable velocity fluctuation in the water cushion. The aeration concentration was measured with a CQ6-2004 aeration concentration meter, and the results are shown in Figure 14.

Aerating a large amount of air at a low-pressure is an effective way to prevent cavitation erosion. When the air content in the water increases, the compressibility of the mixture of water and vapor correspondingly increases, which can mitigate the impact of cavitation collapse and reduce the risk of erosion. Experimental data \[36,37\] showed that when the aeration concentration in the flow reached 1% to 2%, cavitation erosion at the solid boundary was reduced. Moreover, when the aeration concentration reached 5% to 7%, cavitation erosion would not occur. In this paper, the data showed that the aeration concentration ranged approximately from 5% to 25% and decreased with the increasing discharge, which reflected a low probability of cavitation erosion. However, the values obtained in the experiment were almost larger than the results of the numerical simulation, suggesting that the numerical simulation of aeration concentration needs further study. Because of the effects of surface tension and viscosity, the aeration concentration is underestimated so that the aeration concentration is larger in the prototype \[23–26\]. Therefore, the results of aeration concentration could be used as a reference for engineering design.

Figure 14. Aeration concentration in the vortex drop shaft.
4.6. Cavitation Number

The probability of cavitation is lower when there is a greater difference between the absolute pressure and the vapor pressure of a liquid at a certain point. In addition, the larger the velocity is, the more easily cavitation occurs. Therefore, the cavitation number is often used to describe the degree of flow cavitation [38] and can be defined as follows:

\[ N = \frac{p_0 - p_v}{0.5\rho V^2} \]  

where \( p_0 \) is the absolute pressure, \( p_v \) is the elevation-dependent vapor pressure, \( V \) is the resultant velocity close to the wall, and \( \rho \) is the density. The smaller the cavitation number from Equation (14) is, the more easily cavitation occurs.

In hydraulic structures, negative pressure exists in sections of abrupt changes, thereby increasing the probability of cavitation. In this study, the body changed abruptly at the gradient section and the connection between the vertical shaft and outlet tunnel, and therefore cavitation might occur in these sections. The cavitation number was calculated based on the pressure and velocity, and the results are shown in Figure 15.

Generally, cavitation occurs easily in practical applications when the cavitation number is less than 0.2 [39]. In this study, the pressures in the vortex chamber, gradient section and dissipation well were large, and the velocities were small, thus, the associated cavitation numbers were larger than 1.5. Although negative pressure and large velocity appeared in the vertical shaft and resulted in smaller cavitation numbers from 0.40 to 0.80, they were all larger than the critical value of 0.2, so there was a smaller probability of cavitation.

![Figure 15. Cavitation number in the vortex drop shaft.](image-url)
4.7. Energy Dissipation Rate

The energy dissipation within a vortex drop shaft is mainly reflected by two factors. First, the centrifugal force generated by the swirling flow increases the pressure on the wall and consequently increases the frictional resistance. Simultaneously, as the flow swirls along the wall, the trajectory becomes longer and further increases the head loss. Second, the flow fluctuates with aeration and combined with the large velocity gradient, results in swirling, which causes shearing between flow layers. In addition, the shearing, rotation and collision caused by a flow jet that enters the dissipation well can also dissipate energy. In the present study, the calculated section is shown in Figure 1, where the section of volute chamber A-A at $z = 1.9$ m and the section of outlet tunnel B-B at $Y + 1.8$ m. The reference position of the potential energy is $z = 0.0$ m.

The energy dissipation rate of the traditional energy dissipators is generally between 40% and 50%; any rate over 50% is considered excellent. In this paper, the energy dissipation rates exceeded 70% as shown in Table 3, indicating that the vortex drop shaft without ventilation holes sufficiently dissipated energy and met the needs of the actual project. As shown in Figure 16, energy dissipation mainly occurred in the dissipation well and sloped section, and the dissipation rate of turbulent kinetic energy was generally between 20 and 30 $m^2\cdot s^{-3}$ with a maximum of 32 $m^2\cdot s^{-3}$. In addition, the wall friction dissipated some energy, and the dissipation rate of turbulent kinetic energy near the wall in the same cross-section was larger than that in other parts of the structure.

Table 3. The calculation results of the energy dissipation rate.

| Condition | Section A-A | Section B-B | Energy Dissipation Rate (%) |
|-----------|-------------|-------------|----------------------------|
|           | $V^2/2g$ (m) | $H + p/\gamma$ (m) | $V^2/2g$ (m) | $H + p/\gamma$ (m) |               |
| 5%        | Experiment  | 0.16        | 1.65                  | 0.19          | 0.29          | 73.48         |
|           | Simulation  | 0.17        | 1.69                  | 0.20          | 0.28          | 74.12         |
| 2%        | Experiment  | 0.18        | 1.82                  | 0.23          | 0.31          | 73.00         |
|           | Simulation  | 0.19        | 1.81                  | 0.24          | 0.33          | 71.66         |
| 0.1%      | Experiment  | 0.21        | 2.12                  | 0.24          | 0.39          | 72.95         |
|           | Simulation  | 0.25        | 2.13                  | 0.30          | 0.41          | 70.13         |

Notes: $H$ is the elevation head, $\gamma$ is the specific weight of water, $p$ is the gauge pressure.

Figure 16. Dissipation rate of turbulent kinetic energy in the numerical simulation ($P = 0.1\%$).

5. Conclusions

To address the negative pressure and cavitation erosion problems in vortex drop shaft spillways, a new vortex drop shaft without ventilation holes is proposed in this paper. An increased height intake tunnel is used to facilitate aeration. The vortex drop shaft is simulated using the RNG $k-\varepsilon$ turbulence model. By comparing the experimental and simulation results, the hydraulic characteristics of the vortex drop shaft are studied in detail. The results of the research are as follows:
1. The RNG $k$-$\varepsilon$ turbulence model effectively simulated the flow characteristics of the vortex drop shaft. The hydraulic parameters, including the velocity and pressure, agreed well with the experimental data and showed the same trends. Thus, similar swirling problems can be simulated by the RNG $k$-$\varepsilon$ turbulence model.

2. The flow regime in the vertical shaft was stable, but the flow in the dissipation well was swirling and turbulent. The energy dissipation rate exceeded 70% from the gradient section to the outlet tunnel, indicating the occurrence of sufficient energy dissipation. The velocity was small at the top and bottom but large in the middle, and the pressure was large at the top and bottom but small in the middle. Small negative pressure areas were observed.

3. By increasing the clearance height of the intake tunnel and changing the flow conditions, the water flowed into the vertical shaft with a large amount of air. There was a clear phenomenon of aeration and the aeration concentration could provide some reference for engineering design. Additionally, the cavitation number was larger than the critical value, which indicated a low probability of cavitation.

However, our study has some limitations. For example, the diameters of the vertical shaft, vortex chamber and slope section need to be optimized based on existing body sizes to determine the best body shape for safe and stable operation. In addition, considering the scale effect, different scales of experiments and numerical simulations need to be performed.

**Author Contributions:** Conceptualization, W.Z. and J.W.; Methodology, W.Z.; Software, Z.D.; Validation, W.Z., J.W. and C.Z.; Formal Analysis, W.Z.; Investigation, W.Z.; Resources, J.W.; Data Curation, Z.Z.; Writing-Original Draft Preparation, W.Z.; Writing-Review & Editing, W.Z., Z.D. and Z.Z.; Supervision, J.W.

**Funding:** This research received no external funding.

**Acknowledgments:** The numerical calculations in this paper were performed using the supercomputing system at the Supercomputing Center of Wuhan University.

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

**Notations**

$A_s$ surface area (m$^2$)

$C$ aeration concentration

$C_{air}$ coefficient of proportionality

$\varepsilon$ turbulent dissipation rate m$^2$·s$^{-3}$

$F$ volume of fluid (VOF) function

$F_r$ Froude number

$f_i$ gravity component (m·s$^{-2}$)

$g$ gravity acceleration (m·s$^{-2}$)

$g_n$ component of gravity normal to the free surface (m·s$^{-2}$)

$G$ generation of turbulent energy caused by the average velocity gradient

$h$ water depth (m)

$H$ elevation head (m)

$i$ slope

$k$ turbulent kinetic energy (kg·m$^2$·s$^{-2}$)

$L_T$ turbulence length scale (m)

$N$ cavitation number

$P$ flood frequency

$P_d$ disturbance energy per unit volume (N·m$^{-2}$)

$P_t$ stabilising forces per unit volume (N·m$^{-2}$)

$p$ gauge pressure (Pa)

$p_0$ absolute pressure (Pa)

$p_v$ vapor pressure (Pa)

$\rho$ density of water (kg·m$^{-3}$)
Q
radius of the cavity (m)
\( r \)
specific weight of water (N·m\(^{-3}\))
\( \gamma \)
radius of vertical shaft (m)
\( R \)
hydraulic radius (m)
\( R' \)
water trajectory (m)
\( S \)
time (s)
\( t \)
velocity component (m/s)
\( u_i \)
coefficient of dynamic viscosity (kg·m\(^{-1}\)·s\(^{-1}\))
\( \mu \)
revisionary coefficient of dynamic viscosity (kg·m\(^{-1}\)·s\(^{-1}\))
\( \mu_{eff} \)
coefficient of kinematic viscosity (m\(^2\)·s\(^{-1}\))
\( \nu \)
resultant velocity (m·s\(^{-1}\))
\( V \)
tangential velocity (m·s\(^{-1}\))
\( V_t \)
vertical velocity (m·s\(^{-1}\))
\( V_z \)
volume of air entrained per unit time m\(^3\)
\( \delta V \)
coefficient of surface tension (N·m\(^{-1}\))
\( \sigma \)
angle between the velocity direction and the vertical direction
\( \theta \)
factor of friction loss
\( \lambda \)
coordinate component
\( x_i \)

References

1. Lian, J.J.; Yan, M. *Hydrodynamics for High Dam*; China Water & Power Press: Beijing, China, 2008.
2. Zhao, C.H.; Zhu, D.Z.; Sun, S.K.; Liu, Z.P. Experimental study of flow in a vortex drop shaft. *J. Hydraul. Eng.* 2006, 132, 61–68. [CrossRef]
3. Niu, Z.M.; Nan, J.H.; Hong, D. An experimental study on hydraulic characteristics of a novel aerator. *Adv. Water Sci.* 2013, 24, 372–378.
4. Guo, L. *Experimental Study on Bodily Form Optimization of the Vertical Shaft Spillway*; Xi’an University of Technology: Xi’an, China, 2007.
5. Dong, X.L.; Guo, J.; Xiao, B.Y.; Zhou, Z. Design principle of high head and large discharge vortex drop spillway. *J. Hydraul. Eng.* 2000, 31, 27–33.
6. Dong, X.L. *Spillway with Swirling Flow*; The Yellow River Water Conservancy Press: Zhengzhou, China, 2011.
7. Chen, X.W.; Zhang, Z.X.; Chong, L.; Xin, B. Experimental study on pressure of shaft spillway with different depth of stilling well. *Chin. J. Appl. Mech.* 2016, 33, 826–831.
8. Yu, D.Y.; Lee, J.H.W. Hydraulics of tangential vortex intake for urban drainage. *J. Hydraul. Eng.* 2009, 135, 164–174. [CrossRef]
9. Jain, S.C. Tangential vortex-inlet. *J. Hydraul. Eng. ASCE* 1984, 110, 1693–1699. [CrossRef]
10. Del Giudice, G.; Gisonni, C. Vortex dropshaft retrofitting: Case of Naples city (Italy). *J. Hydraul. Res.* 2011, 49, 804–808. [CrossRef]
11. Guo, Y. The hydraulic characteristics of shaft tunnel in vortex flow intake shaft spillway. *J. Hydrodyn* 1995, 2, 146–154.
12. Nan, J.H.; Niu, Z.M.; Zhang, D.; Hong, D. Pressure characteristics of a cavity gyrating flow in gyrating discharge tunnel. *J. Basic Sci. Eng.* 2016, 24, 272–281.
13. Niu, Z.M.; Zhang, M.Y. Basic hydrodynamics characteristics of cavity spiral flow in a large size level pipe. *J. Hydrodyn.* 2005, 17, 503–513.
14. Cao, S.L.; Niu, Z.M.; Yang, J.; Lu, H.B. Velocity and pressure distributions in discharge tunnel of rotary-obstruction composite inner energy dissipation. *Sci. Chin. Technol. Sci.* 2011, 54, 111–117. [CrossRef]
15. Li, Y.; Zhang, F.X.; Deng, J.; Xu, W.L. Calculation of the cavity length of annular aerator in the vortex drop shaft spillway. *J. Sichuan Univ.* 2011, 43, 28–33.
16. Dong, Z.X.; Ping, L.Z.; Zhang, G.J.; Song, W.X. 3-D numerical simulation of flow in shaft spillway with whirling current. *J. Hydraul. Eng.* 2003, 34, 58–63.
17. Gao, X.P.; Zhang, H.; Liu, J.J.; Sun, B.; Tian, Y. Numerical investigation of flow in a vertical pipe inlet/outlet with a horizontal anti-vortex plate: Effect of diversion orifices height and divergence angle. *Eng. Appl. Comput. Fluid Mech.* 2018, 12, 182–194. [CrossRef]

18. Guo, X.L.; Xia, Q.F.; Fu, H.; Yang, K.L.; Li, S.J. Numerical study on flow of newly vortex drop shaft spillway. *J. Hydraul. Eng.* 2016, 47, 733–741.

19. Liu, Z.P.; Guo, X.L.; Xia, Q.F.; Fu, H.; Wang, T.; Dong, X.L. Experimental and numerical investigation of flow in a newly developed vortex drop shaft spillway. *J. Hydraul. Eng.* 2018, 12, 182–194. [CrossRef]

20. Gao, J.Z.; Dong, X.L.; Liu, J.G. Research and application of eco-friendly energy dissipation technology-internal energy dissipation. *J. Hydraul. Eng.* 2008, 39, 1176–1182.

21. Yakhot, V.; Orzag, S.A. Renormalization group analysis of turbulence. *I. Basic Theory. J. Sci. Comput.* 1986, 1, 3–51. [CrossRef] [PubMed]

22. Chen, J.G.; Zhang, J.M.; Xu, W.L.; Wang, Y.R. Scale effects of air-water flows in stilling basin of multi-horizontal submerged jets. *J. Hydrodyn.* 2010, 22, 788–795. [CrossRef]

23. Fuchs, H.; Hager, W.H.; Asce, F. Scale effects of impulse wave run-up and run-over. *J. Waterw. Port Coast. Ocean Eng.* 2012, 138, 303–311. [CrossRef]

24. Pfister, M.; Chanson, H. Scale effects in modelling two-phase air-water flows. In Proceedings of the 35th IAHR World Congress, Cheng Du, China, 8–13 September 2013; pp. 1–10.

25. Chanson, H. Turbulent air–water flows in hydraulic structures: Dynamic similarity and scale effects. *Environ. Fluid Mech.* 2009, 9, 125–142. [CrossRef]

26. Chen, H.Y.; Xu, W.L.; Deng, J.; Niu, Z.P.; Liu, S.J.; Wang, W. Theoretical and experimental studies of hydraulic characteristics of discharge tunnel with vortex drop. *J. Hydrodyn.* 2010, 22, 582–589. [CrossRef]

27. Taghavi, M.; Ghodousi, H. A comparison on discharge coefficients of side and normal weirs with suspended flow load using Flow3D. *Indian J. Sci. Technol.* 2016, 9, 1–11. [CrossRef]

28. Sarkardeh, H.; Zarrati, A.R.; Jabbari, E.; Marosi, M. Numerical simulation and analysis of flow in a reservoir in the presence of vortex. *Eng. Appl. Comput. Fluid Mech.* 2014, 8, 598–608. [CrossRef]

29. Peterka, A.J. The Effect of Entrained Air on Cavitation Pitting. In Proceedings of the Minnesota International Hydraulic Convention, Minneapolis, MN, USA, 1–4 September 1953; pp. 507–518.

30. Šarc, A.; Stepišnik-Perdih, T.; Petkovšek, M.; Dular, M. The issue of cavitation number value in studies of water treatment by hydrodynamic cavitation. *Ultrason. Sonochem.* 2017, 34, 51–59. [CrossRef] [PubMed]

31. Wu, M.J.; Wang, H.Y.; Dai, G.Q.; Yang, Q. Study on the hydraulic characteristics of chute section of high-head bank spillway. *Water Power* 2007, 33, 45–46.

© 2018 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (http://creativecommons.org/licenses/by/4.0/).