Numerical Investigation on Flow Characteristics in the Horizontal Elbow Suction Pipe Connected to Sumps of Large-Scale Pumping Stations

Yetian Zhang\textsuperscript{1*}, Jianjun Feng\textsuperscript{1}, Zhenguo Ge\textsuperscript{1}, Guojun Zhu\textsuperscript{1}, Guangkuan Wu\textsuperscript{1}

\textsuperscript{1}Department of Power Engineering, Institute of Water Resources and Hydro-Electric Engineering, Xi'an University of Technology, Xi'an, Shaanxi, 710048, China
Email: jbzyt@foxmail.com

Abstract. To know the flow characteristics in the horizontal elbow suction pipe connected to sumps of large-scale pumping stations, the distribution of air-entraining vortex and water velocity is studied using ANSYS CFX with considering the gravity. The Volume of Fluid (VOF) method is adopted to track the water level of the sump. The results show that a typical funnel-shaped continuous air-entraining vortex is observed via the air volume fraction. Its size gradually increases from the inlet to outlet of the suction pipe. The pressure changes linearly with the water depth under the effect of gravity whereas the velocity shows non-uniform distribution. The axial velocity distribution of the outlet is non-uniform in the radial direction, which is analogous to the velocity distribution. In addition, the centre wall of the suction pipe outlet may affect remarkably the velocity distribution.

1. Introduction
The forebay and sump of the pumping station are hydraulic construction that link the diversion channel to the inlet pipe of pump unit. Irrational geometry design of sump may cause many adverse effects, i.e., large-scale vortex, unit vibration, cavitation, and even affect the normal operation of the pumping station under some serious cases \cite{1}. Therefore, to provide well inflow conditions for the pump is necessary.

In the early study of forebay or sump in pumping stations, model experiment was the main method \cite{2, 3}. With the development of computer technology, numerical simulations have gradually been widely used in the investigation of the internal flow mechanism of the sump \cite{4-7}. Chuang et al. \cite{8} simulated the free surface of the sump using the VOF model. Compared with the time-averaged and instantaneous free surface, only free surface near pipe wall shows obvious fluctuations and intensifies with the flow rate increasing. Rhee et al. \cite{9} studied the influence of the height of floor cone anti-vortex device installed at the bottom wall of the sump on the internal flow in the sump using numerical simulation. Yamasaki et al. \cite{10} studied the submerged and air-entraining vortices in the sump by using SST $k$-$\omega$ turbulence model and Smagorinsky model based on OpenFOAM. They found that LES is very sensitive to grid. However, Reynolds-averaged Navier-Stokes equations can well predict the air-entraining vortex near the inlet of pipe.

At present, the research on the sump of pumping stations mainly focuses on the vertical pipe, whereas almost no research is studied on flow behaviour of the horizontal elbow suction pipe. And the internal flow mechanism for the sump with suction pipe is not clear. Therefore, it is of great significance to deeply explore the flow characteristics of the sump and the horizontal elbow suction
pipe. In this work, the flow behaviour of horizontal elbow suction pipe and the internal flow in the sump are studied via numerical simulation.

2. Numerical Method

2.1. Calculational Domain
The suction pipe and sump at a large-scale pumping station are adopted as this research object. As shown in figure 1, $D$ is the diameter of the suction pipe inlet, and $H$ is the distance from the free surface to the center of the suction pipe inlet. The relative submerged depth $h_D$ of the suction pipe is the ratio of $H$ to $D$.

![Figure 1. Calculational domain and boundary conditions.](image)

2.2. Meshing
The calculation model is meshed using ANSYS ICEM with a hexahedral structured mesh. To accurately simulate the complex flow near the suction pipe, the mesh of the sump near suction pipe is refined in figure 2. The mesh size in the computational domain is 1,759,440.

![Figure 2. Computational mesh.](image)

2.3. Boundary Conditions
In this work, the ANSYS CFX is adopted for simulating the flow characteristics. The free surface is tracked by the VOF method. There is an obvious interface between air and water in the sump, which is a free surface flow with clear interfaces. The homogeneous flow model is adopted to simulate the gas-liquid two-phase flow. SST $k-\omega$ turbulence model is selected and the gravity is considered. The inlet boundary is set to the height of the free surface using the step function ($h_D = 1.22D$), and the inlet pressure is defined as the static pressure according to the pressure of the water depth distribution in figure 3. The outlet boundary condition is defined as the flow rate ($Q = 800.28 \text{ m}^3/\text{h}$). The top of the sump is set to an opening boundary condition. Figure 1 shows the detailed settings of the boundary conditions.
3. Results and Discussion

The free surface and the air-entraining vortex can be identified by the iso-surface of the air volume fraction, figure 4(a) depicts the structure of air-entraining vortex in the sump and the suction pipe, i.e., the air volume fraction is 0.01. \(Z/D\) is the dimensionless height. As shown in figure 4(b), the vorticity near the air-entraining vortex is significantly larger than that in other regions. The water surface above the inlet of the suction pipe shows an obvious subsidence like a funnel shape. The suction-air extends to the outlet of the suction pipe, which is a typical continuous air-entraining vortex.

As shown in figure 5, \(L\) is the center line of the suction pipe from the inlet to the outlet. Eight slices perpendicular to the centre line are selected to observe the flow field of the suction pipe. The size of air-entraining vortex in the elbow suction pipe gradually increases from the inlet to the outlet.
Figure 5. Air volume fraction distribution of suction pipe.

Figure 6 depicts the distribution of pressure and velocity on the Y=0 slice. The pressure above the water surface in the sump is zero, and the pressure below the water surface in the sump gradually increases along the Z axis due to the influence of gravity. However, the water velocity in the sump is relatively small as a whole with little difference. Obviously, the water velocity near the inlet of the suction pipe is accelerated.

Figure 6. Pressure and velocity distribution of sump.

Figure 7(a) presents the pressure distribution at center line of the suction pipe. Pressure in the range from 1.0L to 0.6L increases gradually, almost unchanging at 0.6L to 0.4L, then decreases gradually. This distribution is mainly related to the change of the suction pipe geometry, which first extends in the -Z axis direction and then in the +Z axis direction. As shown in figure 7 (b), the velocity on the center line generally shows an upward trend with unchanging from 0.9L to 0.4L. The velocity distribution at the pipe inlet is radially non-uniform, whereas that at the outlet shows a non-radial change on the side close to the sump because of the air-entraining vortex. It is called partial non-uniform in the radial direction.

As shown in figure 8 (a), the velocity (V) of outlet is further decomposed in the cylindrical coordinate to obtain the axial velocity (V_a), the circumferential velocity (V_c) and the radial velocity (V_r). The comparison means that the axial velocity distribution at the outlet is highly similar to the velocity distribution, which are non-uniformly distributed in the radial direction. Moreover, the circumferential velocity and radial velocity at the outlet are non-uniformly distributed. As shown in figure 8 (b) and (c), the velocity change trend is similar along the radial direction between the line 1 and line 2. In particular, the velocity components near the center of the outlet vary greatly due to the influence of the center wall.
(a) Pressure distribution of suction pipe.  
(b) Velocity distribution of suction pipe.

**Figure 7.** Pressure and velocity distribution of suction pipe.

(a) Velocity components distribution of outlet in cylindrical coordinate.
(b) Velocity components of line 1.
(c) Velocity components of line 2.

**Figure 8.** Velocity components of outlet.
4. Conclusion
In the work, the flow behavior of the horizontal elbow suction pipe in a large-scale pumping station are studied by numerical simulation. The main conclusions are as follows:

(1) The air-entraining vortex at the inlet of the suction pipe can be accurately identified by the iso-surface of the air volume fraction. When the relative submergence depth \( h_D \) is 1.22\( D \), a typical funnel-shaped continuous air-entraining vortex is observed.

(2) Under the effect of gravity, the pressure distribution in the sump and suction pipe shows a linear change with the water depth. Due to the influence of the geometry, the pressure distribution near the outlet of the suction pipe is non-uniform in the radial direction.

(3) The velocity distribution at the inlet of horizontal elbow suction pipe is non-uniform and that at the outlet is partial non-uniform in the radial direction. The outlet velocity of suction pipe is decomposed in the cylindrical coordinate. The comparison means that the axial velocity \( V_a \) distribution is almost the same as the velocity \( V \) distribution, which are partial non-uniform flow in the radial direction. Besides, the center wall has a major influence on the velocity of the outlet.

Acknowledgement
This research was supported the National Natural Science Foundation of China (Grant Nos. 52079108, 51909212 and 51879216).

References
[1] Zi D, Wang F J, Yao Z F, et al. 2015 Effects analysis on rectifying intake flow field for large scale pumping station with combined diversion piers Transactions of the Chinese Society of Agricultural Engineering (Transactions of the CSAE) 31: 71-77.
[2] Denny D F 1956 An experimental study of air-entraining vortices in pump sumps Proceedings of the Institution of Mechanical Engineers 170(2): 106-116.
[3] Rajendran V P, Constantinescu S G, Patel V C 1999 Experimental validation of numerical model of flow in pump-intake bays Journal of Hydraulic Engineering 125: 1119-1125.
[4] Okamura T, Kamemoto K, Matsui J 2007 CFD prediction and model experiment on suction vortices in pump sump Proceeding of the 9th Asian International Conference on Fluid Machinery.
[5] Kang W T, Yu K H, Lee S Y, et al. 2011 An Investigation of cavitation and suction vortices behavior in pump sump Proceeding of ASME-JSME-KSME 2011 Joint Fluids Engineering Conference: 257-262.
[6] Cong G, Wang F 2007 Numerical investigation on the flow structure and vortex behavior at a large scale pump sump Proceeding of ASME/JSME 2007 5th Joint Fluids Engineering Conference: 947-955.
[7] Zhao Y X, Kim C G, Lee Y H 2013 CFD study on flow characteristics of pump sump and performance analysis of the mixed flow pump IOP Conference Series: Materials Science and Engineering 52: 072006.
[8] Chuang W L, Hsiao S C, Hwang K S 2014 Numerical and experimental study of pump sump flows Mathematical Problems in Engineering 2014 doi.org/10.1155/2014/735416.
[9] Rhee D S, Kim H J 2016 Analysis of flow change in sump by floor cone anti-vortex device height using CFD Journal of Korean Society of Hazard Mitigation 16: 315-321.
[10] Yamasaki R, Kouchi T, Nagata Y, et al. 2017 OpenFOAM analysis of air-entraining and submerged vortices in pump sump Transactions of the JSME (in Japanese).