Effect of suction pipe inlet condition on the occurrence of vortex in pump sump

Ming Guo\(^1\), Zhenmu Chen\(^2\), and Young-Do Choi\(^3\*)

\(^{1,2}\) Graduate School, Department of Mechanical Engineering, Mokpo National University, 1666 Youngsan-Ro, Cheonggye-Myeon, Muan-gun, Jeonnam, 58554, Republic of Korea

\(^3\*)\) Department of Mechanical Engineering, New and Renewable Energy Technology Research, Mokpo National University, 1666 Youngsan-Ro, Cheonggye-Myeon, Muan-gun, Jeonnam, 58554, Republic of Korea

E-mail: ydchoi@mokpo.ac.kr

Abstract: Pump sump is an indispensable facility in irrigation, drainage, agriculture and industrial processes. The water intake conditions play an important role on the performance of the whole system. However, there are different kinds of undesirable vortices usually occurred in pump sump, such as submerged vortex, air entrained vortex. Accompanying with the variety of vortex, noise and vibration are also produced. In this study, a scale down model of pump sump has been designed and constructed. The occurrence of vortex is the concentrated target which was investigated under different suction pipe inlet conditions. Moreover, with respect to the different water levels, flow rate and a fixed distance from the bottom to suction pipe bell mouth detailed experiments were conducted. In addition, a Computational Fluid Dynamics (CFD) analysis method also has been carried out in this paper, aimed at verifying the effect of pipe inlet condition on visualization and formation of vortex which have been obtained by experiment. The comparison of CFD analysis and detailed experiments results were discussed and described.

1. Introduction

The pump sump, for drawing water from a reservoir or river, is widely used in agriculture irrigation, city drainage, and industrial water supply. The performance of pump sump is a key point of the efficiency of a water intake system, which should meet the diverse requirements in the whole system, there are many effect factors need to be considered, such as, the shape, size of water tank, bell mouth, submergence depth, flow rate, and water level etc. Usually, some undesirable vortex occurs in the connection of outlet bell mouth and water supply tank in pump sump, such as submerged vortex or air entrained vortex. If the vortices can’t be suppressed promptly, there will be a greatly negative effect on the water intake system, accompanying with noise and vibration.

Up to now, there have been some investigations on different kinds of vortices in pump sump. Some kinds of standard prototype configurations have been established. For example, TSJ Standard (Turbomachinery Society of Japan) [1], HI Standard of American[2]. To reduce the experimental apparatus costs of pump sump, Matsui.J[3] established a scaling model with relatively high flow rate and set several computational CFD codes of single patterns with different boundary conditions to do the simulation of the water supply system, moreover, CFD analysis results and experimental data...
29th IAHR Symposium on Hydraulic Machinery and Systems  
IOP Conf. Series: Earth and Environmental Science 240 (2019) 062023  
doi:10.1088/1755-1315/240/6/062023

were compared with each other. Cecilia Lucio[4] did the vortex detection to validate the predicting ability of CFD code. Because of the absolute value of total vorticity, it was selected as a useful tool to visualize the different characteristics of concentrated vortex. In addition, A C Bayeuil[5] also used the CFD commercial code to predict the flow behavior in pump sump and studied the flow characteristic of air entrainment by comparing multi-phase numerical study result with experiment investigations. Youngbum Lee[6] investigated the effect of suction leaning angle on the pump sump internal flow. C G Kim[7] examined the flow characteristics around the suction pipe and investigated about the AVD(Anti vortex device) to suppress the submerged vortex.

This study is aimed to investigate the relationship between water levels and flow rate with a fixed distance from the bottom floor to bell mouth, and check the internal flow characteristic in pump sump by detailed experiments in a scale down model and CFD analysis has been conducted using ANSYS CFX[8].

2. Experimental apparatus and numerical method

2.1. Experiment setup

Figure 1 shows the experimental test apparatus of pump sump. The shape of the water tank flow passage is cuboid. The transparent acrylic resin has been used as construction material of water tank, so it is more convenient for visual observation about different kinds of vortex. The bell mouth was set under the water level. The centrifugal pump drives the water from suction pipe to T-connector and releases to the water tank. The straighter in water tank was employed for combing the water flow pattern to make the water flow smoother, and the water turbulence will be smaller. Moreover, the real-time flow rate through the intake pipe can be measured by flow meter in the return circuit. At the end of the pump sump experiment, the water remaining in the water tank flow passage can be drained out through the two drainage taps, so it is convenient to repeat experiment.

Figure 1. The arrangement of experimental apparatus.

Figure 2 shows the plan view and front view of the water tank flow passage model. The size and some parameters of the pump sump experimental apparatus have been presented. The height of water tank is 350mm, the inlet diameter of the bell mouth and suction pipe are $D=80$mm and $d=50$, respectively. The distance from the rear wall to the center line of the suction pipe is 75mm, to more easily induce vortex in the water tank, the location of the suction pipe is not in the center position between the left and right wall. The distance from suction pipe to left wall equal to 85mm, and the distance from suction pipe to left wall is 115mm, $L$ is the distance between the bell mouth of suction pipe and the bottom floor of water tank, $Q$ is the flow rate for experiment centrifugal pump, $H$ is the water level in water tank, the three factors have a great effect on the flow pattern and characteristics of pump sump, firstly, $Q$ and $H$ were fixed, and $L$ have been investigated by several detailed experiments.
with changing the height of $L$. In this study, quantitative analysis was conducted to investigate the occurrence of vortex.

A series of experiments were conducted to find out the relationship between vortex occurrence and flow rate, water level. In this step, the critical value of water level for air entrained vortex and submerged vortex occurring with different flow rate was investigated. Table 1 shows the different flow rate which have been set in detailed experiment, $L$ is fixed as 30mm. For the method of implementation experiment, at first, fill the tank with water to maintain a high water level in the tank. Then keep the drainage tap of water tank open to drain water away slowly, so the water level is decreased with water drainage, meanwhile, the water level in water tank is also monitored. When air entrained vortex occurred for the first time, the corresponding water level is recorded and the drainage tap is closed immediately, then observing the vortex for ten minutes to confirm the occurrence of vortex. After that, with drainage of water continuously, the water level is recorded again when the submerged vortex occurs.

**Figure 2.** Shape and size of water tank flow passage model.

| $Q$(m$^3$/h) | 2.55 | 3.52 | 4.45 | 5.51 | 6.62 | 7.43 | 8.32 | 9.01 |
|-------------|------|------|------|------|------|------|------|------|

2.2. Numerical method

In order to identify the occurrence of visible vortex, the flow pattern and vortex characteristics have been investigated by numerical method, 3-D model was also established which have the same shape and size with pump sump experimental apparatus. The finite volume method of ANSYS CFX code was adopted to do the numerical simulation. Multi-phase flow in the water tank was simulated. Matsui,J.[3] have presented different CFD simulation cases with tetrahedral mesh and hexahedral mesh by different CFD software. Regarding his result of CFD simulation, hexahedral mesh have a better display of water surface and vortex visualization. Therefore, for the numerical grid, hexahedral mesh have been applied in all the flow field as shown in Figure 3. In order to ensure a more precise simulated result, The average value of $y^+$ of full fluid domain is below 33. the mesh around the bell mouth has been refined, and a grid number of about 1.3×10$^6$ elements was employed of whole fluid domain.

The boundary condition of CFD analysis is shown in Figure 4,The same with case 7 of experiment. Unsteady state has been employed to do CFD analysis. water mass flow rate of 8.32m$^3$/h was set in inlet and outlet, there is no speed setting in the air above water level, air flows freely. The boundary condition of opening was set for the upper surface of the flow passage. No-slip wall condition was
applied to the pump sump wall region and unsteady state calculation was conducted, the two equation turbulence of SST (shear stress transport) was adopted in numerical calculation.

![Figure 3. Numerical grid of fluid domain.](image)

![Figure 4. Boundary condition and coordinates.](image)

3. Result and discussion

3.1. Experiment result

Figure 5 presents the TSJ standard of vortex classification in pump sump, surface dimple vortex, intermittent vortex, and continuous vortex belong to air entrained vortex. The appearance of different kinds of vortex usually changes a lot, and sometimes some of them will exist at the same time. The surface dimple vortex is the easiest to generate, usually the location of dimple vortex is near the water surface around the suction bell mouth, then it is intermittent vortex. Intermittent vortex and continuous vortex can be converted into each other, the submerged vortex is the most difficult to produce. and it needs to meet many requirements at the same time. During the process of experiment, when the submerged vortex occurred, the water surface around the bell mouth was very unstable and changed drastically. The intensity of vortex is affected by many factors. The interval between vortex generation and disappearance is also a important factor, more investigation will be conducted in the future work.
**Figure 5.** Vortex classification. [1]

**Table 2.** Critical value of water level (Time averaged value).

| Q (m³/h) | 2.55 | 3.52 | 4.45 | 5.51 | 6.62 | 7.43 | 8.32 | 9.01 |
|----------|------|------|------|------|------|------|------|------|
| H₁ - Air entrained vortex (mm) | 55 | 61.5 | 68 | 75 | 79 | 86 | 90 | 95.5 |
| H₂ - submerged vortex (mm) | - | - | 37.5 | 40 | 43.5 | 46.5 | 50.5 | 54.5 |

**Figure 6.** Air entrained vortex (surface dimple vortex) observation.

Figure 6 shows that the surface dimple vortex occurs near the water surface, when it occurs, it has a little effect on the fluctuation of water surface, and it also disappears and appears irregularly. Figure 7 shows the occurrence of submerged vortex under the water surface. In order to be more accurate, as soon as the vortex appears, the critical value is recorded immediately. For the occurrence of submerged vortex, there are always air entrained vortex accompanying, vibrations and noise are also generated. All of the recorded critical values are shown in Table 2. As the height of L was set as
30mm, and when submerged vortex occurs, the water surface is unstable, the critical water level value at the flow rate of 4.45m$^3$/h is 37.5mm, it means that the bell mouth is under the water level only 7.5mm, if the water level is lower, air may be inhaled into the suction pipe and centrifugal pump may be damaged. So in the series of experiment of vortex occurrence, six cases was conducted, for dimple vortex, there are eight cases. For the data recorded during the process of experiment, which are shown in Table 2 in details. $H_1$ and $H_2$ are the critical water level of dimple vortex and submerged vortex, respectively.

![Submerged vortex visualization](image1.png)

**Figure 7.** Submerged vortex visualization.

![Regional division of vortex occurrence](image2.png)

**Figure 8.** Regional division of vortex occurrence.

Figure 8. shows the nearly linear relationship between water level and flow rate, the distance from bell mouth to the bottom of water tank was set as 30mm, as shown in the figure, the first quadrant of the
coordinate region is divided into 3 areas. Area (1) represents that no vortex occurs in this region. In Area (2) and (3), there are several kinds of air entrained vortex occurring. However, the submerged vortex only occurs in Area (3). For the occurrence of submerged vortex, it needs higher flow rate and lower water level.

3.2. CFD analysis result
To verify the occurrence of visible submerged vortex and examine the formation and behaviour of the submerged vortex, water level has been set as the recorded critical value when submerged vortex occurred. Numerical simulation with two-phase flow has been performed by numerical code of ANSYS CFX. A cut plane was set in the bell mouth of suction pipe, moreover, the diameter of contour circle equal to the diameter of suction pipe bell mouth. As illustrated in Figure 9(a). Approximately in the center of the bell mouth, a Iso-surface of equal vorticity has been established which is stained by velocity v by means of the described visualization of CFD software. The colour changes represent that the velocity increases from bottom to inner of the suction pipe. Figure 9(b). Figure 9(c) and Figure 9(d) present the computational result of water velocity distribution, water velocity swirling strength distribution and pressure distribution, respectively. In the position of vortex occurrence, there are higher water velocity, higher water velocity swirling strength and lower pressure distribution. To check the internal flow behaviour and characteristic, the distribution of component velocity on Z axis has been investigated which is shown in Figure 10. The coordinate origin is located in the center of suction pipe bell mouth which has shown in Figure 4. There is a large negative velocity component of \( v_x \) in the center of bell mouth. For \( v_z \), the absolute value is symmetrical distributed approximately, half of it is negative, and another half is positive. Therefore, water around the vortex core can rotate, the vortex can be produced.

Figure 9. Characteristics of internal flow.

Figure 10. Velocity component distribution.

4. Conclusion
In this study, a series of experiment have been conducted to investigate about the suction pipe inlet condition, the dimple vortex and submerged vortex critical value of water level in different flow rates
have been recorded in detailed experiment, the approximate linear relationship between water level and flow rate has been verified in a fixed distance from bell mouth to bottom floor.

Numerical simulation has also been carried out to investigate the behaviour and characteristics of internal flow in ANSYS CFX. A vortex core has been obtained, water velocity and water velocity swirling strength distribution of vortex core region is higher, and pressure is lower than other regions. These are effective factors for the occurrence of submerged vortex in pump sump.

References

[1] Turbomachinery Society of Japan 2005 Standard Method for Model Testing the Performance of a Pump Sump, TSJ S002
[2] Hydraulic Institute Standards 1998 American National Standard for Pump Intake Design, ANSI/ASME 1.9.
[3] Matsui J, Kamemoto K and Okamura T 2006 CFD Benchmark and a Model Experiment on the Flow in a Pump Sump. Proc. of 23rd IAHR XXIX Symp., Yokohama, Japan
[4] Cecilia L, Sergio L G D 2010 Vortex Detection in Pump Sumps by Means of CFD. Proc. of 25th IAHR Symp., Punta Del Este, Uruguay
[5] Bayeul A C, Simonet S and Bois G, Issa A 2012 Two-Phase Numerical Study of the Flow Field Formed in Water Pump Sump: Influence of Air Entrainment. Proc. of 26th IAHR XXIX Symp., Lille, France
[6] Lee Y, Kim K Y, Chen Z M and Choi Y D 2015 The Effect of Suction Pipe Leaning Angle on the Internal Flow of Pump Sump. Journal of the Korean Society of Marine Engineering 39(8) 849–55
[7] Kim C G, Choi Y D, Choi J W and Lee Y H 2012 A Study on the Effectiveness of an Anti Vortex Device in the Sump Model by Experiment and CFD. Proc. of 26th IAHR XXIX Symp., Lille, France
[8] ANSYS Inc., ANSYS CFX Documentation, Ver. 18.1, http://www.ansys.com