Experimental and CFD analysis for prediction of vortex and swirl angle in the pump sump station model

C G Kim¹, B H Kim¹, B H Bang² and Y H Lee³,⁴

¹ Department of Mechanical Engineering, Korea Maritime and Ocean University, 727 Taejong-ro, Youngdo-gu, Busan 606-791, Republic of Korea
² Daewoo Institute of Construction Technology, 60, Songjuk-dong, Jangan-gu, Suwon-si, Gyeonggi-do 440-800, Republic of Korea
³ Department of Mechanical and Energy system Engineering, Korea Maritime and Ocean University, 727 Taejong-ro, Youngdo-gu, Busan 606-791, Republic of Korea

E-mail: lyh@kmou.ac.kr

Abstract. Sump model testing is mainly used to check flow conditions around the intake structure. In present paper, numerical simulation with SST turbulence model for a scaled sump model was carried out with air entrainment and two phases for prediction of locations of vortex generation. The sump model used for the CFD and experimental analysis was scaled down by a ratio of 1:10. The experiment was performed in Korea Maritime and Ocean University (KMOU) and the flow conditions around pump's intake structure were investigated. In this study, uniformity of flow distribution in the pump intake channel was examined to find out the specific causes of vortex occurrence. Furthermore, the effectiveness of an Anti Vortex Device (AVD) to suppress the vortex occurrence in a single intake pump sump model was examined. CFD and experimental analysis carried out with and without AVDs produced very similar results. Without the AVDs, the maximum swirl angle obtained for experimental and CFD analysis were 10.9 and 11.3 degree respectively. Similarly, with AVDs, the maximum swirl angle obtained for experimental and CFD analysis was 2.7 and 0.2 degree respectively. So, with reference to the ANSI/HI 98 standard that permits a maximum swirl angle of 5 degree, the use of AVDs in experimental and CFD analysis produced very desirable results which is well within the limit.

1. Introduction

It is understood that quite a number of problems faced in a pumping station are related to the design of sump or intake rather than pump design. As an important part of the pumping station, sump is designed to provide uniform, swirl and entrained air free flow to the pump intake. Undesirable flow conditions (entrained air, non-uniform flow distribution, vortices, etc.) can reduce the pump efficiency; induce vibration, noise, cavitation, lead to excessive bearing loads of impeller and structure damage. Due to the quite complex nature of the flow, although the flow conditions causing pump problems are well established, there are no specific solutions to eliminate them. There are some design guidelines for the specific geometrical and hydraulic constraints of the pump sump, for any project, the model study is the only tool for solving potential problems in new designs and rectifying problems observed in existing installations.

⁴ Corresponding author.

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.

Published under licence by IOP Publishing Ltd
Rajendran et al.[1] have made numerical analysis for the flow characteristics of a sump model with pump intake and good agreements were achieved by comparing the numerical results with the experiments. Iwano et al.[2] have introduced a numerical method for the submerged vortex by analyzing the flow in the pump sump with and without baffle plates. Lee et al.[3] have conducted the CFD analysis of a multi-intake pump sump model to check the flow uniformity by predicting the location, number and vorticity of the vortex. The cavitation phenomena has been studied by various investigators. Van et al.[4] have investigated the cavitation inception behavior of a mixed-flow pump impellers model and the calculated results matched well with the experiments. Okamura et al.[5] have evaluated two numerical cavitation prediction methods used in the pump industry manufacturers, and attempts were made to improve the cavitation performance. Li et al.[6] have conducted the numerical analysis on the basis of the development liquid/vapor interface tracking method to predict the cavitation characteristics with a centrifugal pump impeller model.

Experiments were conducted in using a sump station model scaled down by a factor of 10. An Anti-vortex device (AVD) was designed and the experiments were conducted with and without AVDs. Numerical simulations were done on the same model and the experimental results and the numerical results were compared on the basis of swirl angle and vortex.

2. Sump model

Table 1 shows the various dimensions of the model parameters both real scale model and the model used for the test. The design was adopted from the guidelines dictated by ANSI/HI 9.8[8] standard. It consists of 2 drum screens, 2 inlets, 3 pumps and one dummy channel. The actual sump station length and width are 65,800 mm and 34,000 mm respectively whereas the model test size is 6,580 mm long and 1,700 mm wide.

| Model Parameters                  | Real scale model | Model test   |
|-----------------------------------|------------------|--------------|
| Bell throat diameter(d)            | 900 mm           | 90 mm        |
| Bell mouth diameter(D)             | 2,000 mm         | 200 mm       |
| Bell mouth ground Clearance        | 1,300 mm         | 130 mm       |
| Rated flow rate                    | 18,000 m³/hr     | 56.9 m³/hr   |
| Rated flow vel.(bell mouth)        | 1.203 m/s        | 0.381 m/s    |
| Sump station length                | 65,800 mm        | 6,580 mm     |
| Sump station width                 | 34,000 mm        | 1,700 mm     |

3. Experimental setup

Figure 1(a) shows the overall experimental setup in which the water is circulated through 3 channels with the help of 55 kW pump. An inverter is implied to control the rpm of the pump and hence the mass flow rate. The pump operates at 8 m head and the maximum flow rate attainable is 900 m³/h. Electro-magnetic flow meters are installed in each channel. Flow meters measurement range varies from 0.3 m/s- 10 m/s with accuracy of F.S±0.5%. Figure 1(b) shows the front channel and bell mouth made from acrylic resin such that the flow pattern and vortex formation can be easily observed.

A four blade zero-pitch swirl meter, which is supported by a low friction bearing, is installed at about four suction bell throat pipe diameters downstream from the pump suction to measure swirl angle of pump approaching flow. For the swirl meter, the tip to tip blade diameter is 0.75d and the length in flow direction is 0.6d and one of the four blades is painted yellow as a reference to count revolutions. Measuring tape is attached in front of the acrylic to determine the water level. Wire mesh used inside the sump with drum screen has same porosity as the real case.
Several attempts were made to finalise the distributors and the bottom AVD for this design. Figure 2 depicts the AVD design and the distributors. Various distributors combination were tried to ensure the uniform flow inside the channel. Similarly curtain walls were implied to reduce the free surface vortices and bottom AVD to remove the submerged vortices. A through investigation was made to ensure the optimum flow and swirl angle. As ANSI/HI 9.8 standard states the maximum swirl angle allowed is 5°, several combinations were tried to obtain the same. 3 distributors, 1 bottom AVD and 1 curtain wall is implied in each channel for the experimentation. Figure 2(b), (c) shows the bottom AVD and the distributors location.

![Experimental setup](image1)

![Bell mouth](image2)

**Figure 1.** Experimental Model setup.

3.1. Test Condition

Table 2 summaries the overall operating conditions of pump station. 3 cases of experiments are listed in the table. Each pump operates with the rated flow rate of 56.9 m³/hr. For the LR-1 case, all the three pumps are in operation with both stop logs and both inlet gates open. For the LR-5 case, pump 1 does not operate (pumps 2 and 3 operating) with stop log 1 and inlet gate 1 in closed position. For the LR-6 case, pumps 1 and 2 are in operation (pump 3 does not operate) with stop log 2 and inlet gate 2 in closed position. Each x shown in the table shows the location of closed positions/pumps.

3.2. Swirl angle Calculation

Swirl angle predicts the intensity of flow rotation. Swirl angle is calculated by experimental method according to HI standard and this is used to check the flow rotation in the suction pipe. The observation time for the swirl meter rotation was above 10 minutes. The HI recommended criterion for swirl angle is within 5 degrees. Equation (1) is experiment calculation method with swirl meter. The swirl angle $\theta$ is defined as follows:

$$\theta = \tan^{-1} \left( \frac{V_x}{V_z} \right)$$

\[ (1) \]
where $V_z = \text{axial mean velocity} = \frac{4Q}{\pi d^2}$, $V_t = \text{tangential mean velocity at the swirl meter} = \frac{\pi dn}{60}$, $Q = \text{flow rate (m}^3/\text{s})$, $d = \text{bell throat diameter (m)}$, $n = \text{revolution of swirl meter (rpm)}$.

For the swirl angle calculating method with CFD, the key point is to obtain the average tangential velocity, so the swirl check circles were created. But for numerical method, the swirl check circle is located at the section of 4d height with diameter of 0.25d, 0.5d and 0.75d respectively. Swirl angle $\theta$ is defined as follows:

$$V_\theta = \frac{\sum_{i=1}^{N} V_{0.25i}}{N} + \frac{\sum_{i=1}^{N} V_{0.5i}}{N} + \frac{\sum_{i=1}^{N} V_{0.75i}}{N} \times 2$$

where $V = \text{circumferential velocity of check point in check circle, respectively (m/s)}$, $N = \text{check point total numbers in single check circle}$.

### Table 2. Operating conditions of pump station.

| Case | Pump operation | Stop log | Inlet gate | Operating condition |
|------|----------------|----------|------------|---------------------|
| LR-1 | #1 #2 #3       | o o o o  |            |                     |
|      | #2             | x o o x  |            |                     |
| LR-5 | #1 #2 #3       |          |            |                     |
|      | #1 #2 #3       | x o x x  |            |                     |
| LR-6 | #1 #2 x        | o x o x  |            |                     |

### 4. Numerical method

A structured hexahedral mesh was generated using ICEM CFD. Initially, the difference between a single phase calculation and a two phase calculation was done. In the one phase numerical analysis, there is no noticeable effect in pressure at the region behind the distributor when it is inserted. But in two phase analysis, there is a pressure drop behind distributor and its effect has to be considered. Therefore, for all numerical cases, a two phase steady state calculation was selected. Therefore, the numerical analysis was carried using a finite volume solver ANSYS CFX v.13.0. The Shear Stress Transport (SST) turbulence model was selected to model the turbulence in the numerical model with a total 9000000 after mesh sensitivity testing.

### 5. Results

#### 5.1. Experimental and CFD results comparison

Table 3 shows the comparison between the experimental and CFD results without the AVD installed for each of the cases. Currently, numerical simulations have only been able to predict the location of the formation of vortices but are unable to determine the strength of the vortex. The CFD result shows good prediction of the location of the vortex. In addition, the swirl angles are also compared and show that the CFD result predicts high swirl angles and in some cases almost accurately.

Table 4 shows the results when the model has the AVD installed. Both results show that no vortices form and the swirl angle have significantly reduced.
The results from both tables are plotted in figure 4 with the HI standard guide line highlighted at a swirl angle of 5 degrees. The differences in the results can be due to some unavoidable errors in experiments, such as the water flow causing the swirl meter to fluctuate and not completely turn, and the innate problem that CFD cannot exactly replicate experimental conditions. In spite of this, the CFD results follow the trend of the experimental results closely. Predicting swirl angles above the standard when the AVD is not installed and the reduction in swirl angle when the AVD installed. The CFD results overall predict are more stable flow due to the fact that the calculations are done in steady state.

### Table 3. Comparison of vortex region and swirl angle without AVD by CFD and experiment.

| Case  | Scale | Pump       | CFD vortex region | Test vortex region | Swirl angle |
|-------|-------|------------|-------------------|--------------------|-------------|
|       |       |            |                   |                    | CFD         |
| LR-1  | 1:10  | #1         | LRSW a, BW b, B c | RSW, BW, B         | 10.7        |
|       |       | #2         | LRSW, BW, B       | LRSW, BW, B        | 7.6         |
|       |       | #3         | LRSW, BW, B       | N/A                | 3.9         |
| LR-5  | 1:10  | #2         | LSW, BW, B        | LRSW, BW, B        | 7.2         |
|       |       | #3         | LSW, B            | N/A                | 11.3        |
| LR-6  | 1:10  | #1         | LSW, B            | RSW, B             | 3.3         |
|       |       | #2         | LRSW, B           | LRSW, B            | 6.7         |

a Left right side wall vortex;  b Back wall vortex; c Bottom vortex; d Free surface vortex

### Table 4. Comparison of vortex region and swirl angle with AVD by CFD and experiment.

| Case  | Scale | Pump | CFD vortex type | Test vortex type | Swirl Angle |
|-------|-------|------|-----------------|------------------|-------------|
|       |       |      |                 |                  | Test         | CFD          |
|       |       |      |                 |                  | degree(°)    | degree(°)    |
| LR-1  |       | #1   | NV              | NV               | 2.7          | 0.07         |
|       |       | #2   | NV              | NV               | 0.9          | 0.19         |
|       |       | #3   | NV              | N/A              | N/A          | 0.04         |
| LR-5  | 1:10  | #2   | NV              | NV               | 1.9          | 0.23         |
|       |       | #3   | NV              | N/A              | N/A          | 0.08         |
| LR-6  |       | #1   | NV              | NV               | 1.4          | 0.1          |
|       |       | #2   | NV              | NV               | 1.1          | 0.08         |

Figure 5 depicts the velocity contours and vortex core region. Figure 5(a) shows the velocity contours at the plane 1 meter above the bottom without an AVD and Figure 5(b) shows the same with an AVD. While comparing the two figures, it is observed that there is a strong jet flow in the common bay in figure 5(a) and so a high swirl angles develop. But in figure 5(b), the distributors help to make the flow uniform, reduce the axial flow and hence results in a very small swirl angle. Figure 5(c) shows the vortex core region obtained by submerged bottom vortex method. The side walls (left and right), back wall and bottom vortex are show in the figure with red circles.
6. Conclusions

Experiments were conducted in dedicated sump model installed in the laboratory of KMOU and numerical simulations were performed in ANSYS CFX 13.0. Using these two methods the prediction of vortex and swirl angle were performed from the obtained results with and without AVD. This paper can be concluded with the following points.

1) In an initial investigation, the effect of the distributor is correctly predicted and the pressure drop across it can be seen in two-phase calculation whereas one phase calculation can lead to incorrect predictions near the bell mouth almost negating the effect of AVD.

2) CFD results show good prediction of the location of vortex and the swirl angles comparison find good agreement with one another. Side walls (left and right), back wall and bottom vortices i.e. all submerged vortices are also characterised.

3) Strong jet flow and high swirl angle is observed in common bay without the use of an AVD. This can be controlled by using an AVD, thus making the flow uniform and reducing the axial flow. The vortex is suppressed and eliminated and the swirl angle is reduced with the use of an AVD.

4) The differences in the results can be due to some unavoidable errors in experiments, such as the water flow causing the swirl meter to fluctuate and not completely turn, and the innate problem that CFD cannot exactly replicate experimental conditions. In spite of this, the CFD results follow the trend of the experimental results closely.
Acknowledgement
This Research was supported by the Daewoo Institute of Construction Technology for Optimization of Sump Station Project, South Korea.

References
[1] Rajendran V P, Constantinescu S G and Patel V C 1998 Experiments on Flow in Model Water-pump Intake Sump to Validate a Numerical Model Proc. of ASME Fluids Eng. Division Summer Meeting (Washington, DC, USA, 21-25 June 1998) 98-5098
[2] Iwano R, Shibata T, Nagahara T and Okamura T 2002 Numerical Prediction Method of a Submerged Vortex and Its Application to the Flow in Pump Sumps with and without a Baffle Plate Proc. of the 9th Int. Symp. on Transport Phenomena and Dynamics of Rotating Machinery (Honolulu, Hawaii, USA, 10-14 Feb 2002) pp 1-6
[3] Choi J W, Choi Y D, Kim C G and Lee Y H 2010 J. Mech. Sci. and Tech. 24(7) 1389-400
[4] van Os M J, Op de woerd J G H and Jonker J B 1997 A parametric study of the cavitation inception behavior of a mixed-flow pump impeller using a three-dimensional potential flow model Proc. of ASME Fluids Eng. Division Summer Meeting (Vancouver, Canada, 22-26 June 1997) 97-3374
[5] Dupont P and Okamura T 2003 Int. J. Rotating Machinery 9(3) 163-70
[6] Li J, Liu L J, Li G J and Feng Z P 2007 J. Eng. Thermophysics 28(6) 948-50
[7] Kim C G, Choi Y D and Lee Y H 2012 Earth and Environment Science IOP IAHR XXVI-083
[8] Hydraulic Institute 2012 American National Standard for Pump Intake Design ANSI/HI 9.8