Numerical Analysis of Vortices Behavior in a Pump Sump

Ayham Amin 1, Byung Ha Kim2, Chang Gu Kim3 and Young Ho Lee4

1, 2 Department of Mechanical Engineering, Korea Maritime and Ocean University, Busan, South Korea
3 Brain Korea 21 Plus, Korea Maritime and Ocean University, Busan, South Korea
4 Divisions of Mechanical Engineering, Korea Maritime and Ocean University, Busan, South Korea

Abstract. A Pump sump or intake is a hydraulic structure with specific dimensions used to contain the water which has to be pumped into the piping system. Recently, the designers and engineers of pump station realized that the efficiency of pump station largely depends on sump structure design, not only on the performance of the selected pump. However, the poor intake design may result in many undesirable phenomena such as a form of swirl and vortices, which lead to affect the pump efficiency. In this study, both numerical and experimental analysis of a rectangular pump sump was carried out to predict swirl angle and free surface vortices formation. The swirl angle experimentally was obtained using a traditional swirl meter installed at the suction pipe in rectangular single pump intake. The objective was to obtain the average tangential velocity at different suction pipe diameter. For free surface vortices the objective was to observe the different vortex formation, air entrance and location. The swirl angle and average tangential velocity estimated by CFD simulation was in agreement with experimental results. The four types of free surface vortices (Surface swirl, Surface dimple, air bubbles and full air core to intake) was also observed clearly in ANSYS CFX.

Keywords: Pump sump • Free surface vortices • Swirl angle • Computational fluid dynamics (CFD)

1. Introduction
A pump sump or intake is simply defined as a hydraulic structure with specific dimensions used to contain the water which has to be pumped into the piping system. Ideally, the flow of water into any sump or intake should be uniform, steady and free from swirl, vortex, and entrained air. Most recurring problems faced in a pumping station are related to the sump design or intake design rather than pump design. There are some international design guidelines specifying geometrical and hydraulic constraints of pump sump. Following these standards restrict undesirable flow patterns up to a certain extent. The implication of which does not guarantee a problem free sump but provides a basis for initial design. Therefore, a sump model test is necessary to check the flow condition around intake structure [1-2].
Swirl angle predicts the intensity of flow rotation. It is considered as an important parameter to define the flow quality ingested by the intake [3]. ANSI/HI 9.8 specify the criteria needed to estimate this parameter. HI recommends the swirl angle to be within 5 degrees. However, in the present work both experimentally and numerically the swirl angle is estimated without any anti-vortex device. As such the values obtained in the present work is slightly above the standard value. The vortices in vicinity of pump intakes may be adjacent to the channel bottom or channel wall (i.e. submerged vortices) or may appear adjacent to the free surface (i.e. Free surface vortex), according to the free surface vortices from ref [4], to evaluate the strength of vortices at pump intakes systematically, the vortex scale varying from surface swirl or dimple to an air core vortex, as shown in Fig. 1. Sometimes an additional device may be installed in the form of floor splitters, distributor, surface beams etc. It is aimed to control the vortex and swirl formation. For empirical vortex data, there is still no simple method for estimating the circulation and vortices generated by non-uniform approach flow. From various experimental data, correlation have been derived which can be helpful for quantitative assessment of where vortices may present problem [5].

Many researchers studied the flow conditions and undesirable phenomenon in sump intake. Luca Cristofano et al. [6] made an experimental study on unstable free surface vortex and gas entrainment onset condition. The main purpose of his research was to understand the influence of different parameters on free surface vortex formation and evolution. Lee et al. [7] conducted numerical analysis for the flow characteristic around pump intake by the unbalanced velocity in the direction of flow passage. Kim et al. [8] made comparative analysis of flow behavior around intake entrance by PIV method and CFD analysis, also he studied a swirl angle of swirl meter at different types and location. Iwano et al [9] introduced numerical method for the submerged vortex by analyzing the flow in the pump sump with and without baffles plates. In our study numerical and experimental analysis have been used to study the flow field in pump intake. The main purpose of this work is to check swirl angle at different cases, calculate average tangential velocity at different suction parameter and predict the free surface vortex location, types and air entrainment.

2. Sump Design and Experimental Setup

Shown in Fig 2 is the three dimensional models of the geometry for sump fluid domains that were made using NX 6.0 according to the design standards ANSI/HI 9.8. It consists of one rectangular channel, Non uniform flow distributor and traditional swirl meter. The width of intake channel W is 500 mm and the length L is 4000 mm, the center of suction bell is located at 200 mm (B), 250 mm (L1) and 100 mm from rear wall, side wall and bottom respectively. The water level (H) is 0.8 m. The gaps between each column in flow distributer increasing from 10 mm up to 55 mm. The aim of
distributor is to generate a non-uniform flow which helps to control and prediction the free surface vortex types and strength.

2.1. Experimental Setup
The experimental setup at Korea Maritime and Ocean University is shown in Fig. 3. The water is circulated through one channel with the help of a 55kW pump. An inverter is used to control the pump rpm and hence the mass flow rate. A four blade zero-pitch swirl meter which is supported by a low friction bearing, installed at about four suction pipe diameter (d) downstream from the pump suction is used 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 revolution. The swirl angle analysis and observation time of swirl meter should be a continuous period of time for comparison. Five cases have been studied; the cases sequence started from the maximum flow rate 150.5 $(m^3/h)$ and, afterward the flow was decreased to minimum value 30.7 $(m^3/h)$.
A traditional swirl meter used to predict the flow rotation by swirl angle calculation. The swirl angle is calculated according to HI standard. The observation time for the swirl meter rotation was above 10 minutes. Equation (1) is experiment calculation method used to calculate the Swirl angle.

$$\theta = \tan^{-1} \frac{V_\theta}{V_z}$$

(1)

Where $V_z = \text{Axial mean velocity} = \frac{4Q}{\pi d^2}$

$V_\theta = \text{Tangential mean velocity at swirl meter} = \frac{\pi dn}{60}$

$Q = \text{flow rate } m^3/h$

$d = \text{bell mouth diameter (m)}$

$n = \text{revolution of swirl meter (rpm)}$

![Figure 4. Swirl meter location](image)

3. Numerical Method

For numerical simulation, commercial CFD code of ANSYS CFX 13.0 [4] is adopted. As boundary condition for transient calculation, the top of the air domain is assigned as opening condition with absolute pressure equal to atmospheric pressure. Both phases (air and water) were distinctly defined by giving initial volume fraction as either one or zero. Initial free surface was the interface of the sump water domain and air domain. The swirl meter was set as a rigid body with free rotation about Z axis. ICEM CFD was used to generate unstructured hexahedral mesh of sump fluid domain. The flow domain was divided into 5 smaller regions to improve the mesh quality. For modeling of turbulence, SST model is adopted. Total number of mesh nodes is 1,434,934. Two-equation turbulence models are widely used to provide a ‘closure’ to the time averaged Navier-Stokes equations. Two principle closure models exist commercially, which are the $k - \varepsilon$ ($k$-epsilon) model and the shear stress transport (SST) model. The two equation models are much more sophisticated than the zero equation models. In these equations both the velocity and the length scale are solved using separate transport equations. The details regarding these aspects are not considered here. These details are available in ANSYS 13 user guide [10].

The CFD calculation was conducted to predict the swirl angle. For calculating the swirl angle in CFD, the key point is to obtain the average tangential velocity. So the swirl check circles were created as shown in Fig.5. The swirl check circle is located at the section of 4d height with diameter of 0.25d,
0.5d and 0.75d respectively. The average axial velocity at swirl meter is defined as given by equation (2) below:

\[
V_a = \frac{\sum V_{x_2} / N + \sum V_{x_3} / N + \sum V_{x_4} / N}{3} \times \chi
\]

Where \( V_a \) = Flow rotation speed at swirl meter (m/s).
\( V \) = circumferential velocity (m/s).
\( N \) = check point total numbers in single check circle.

\[\text{Figure 5. Swirl check circles (CFD)}\]

The ANSYS CFX Solver module was used to obtain the solution of the CFD analysis. The solver control parameters were specified in the form of solution scheme and convergence criteria, high resolution was specified for the solution while for the convergence the residual target for RMS value was specified as 1x10^-5. Table 1 shows the other CFX pre solution setting.

| setting          | Choice               | setting         | Choice                      |
|------------------|----------------------|-----------------|-----------------------------|
| Inlet opening    | Opening pressure     | Free surface    | Standard (mixture model     |
|                  | pressure             | model           | homogeneous)                |
| Outlet           | Normal speed         | Turbulence      | SST                         |
| Total time       | 15 s                 | Interfaces      | 4                           |
| Time step        | 0.01                 | Swirl meter     | Free rotation               |
| Fluid            | Air/water            | wall            | No slip condition           |

4. Results and Discussion
Table 2. shows the swirl angle results from CFD calculations and experiment. For experiment the rotation of swirl meter is recorded over 10 minutes. As shown in Fig 6 the two ways used to estimate the swirl angle by CFD method showed good agreement with experimental results.
Table 2. Swirl angle calculation results (Exp and CFD)

| Case No | Pump operation. (m³/h) | Average velocity (m/s) | Exp (deg) | CFD-Rigid body* | CFD-tangential velocity* (deg) |
|---------|------------------------|------------------------|-----------|----------------|-----------------------------|
| 1       | 30.7                   | 0.07                   | 15.5      | 13.1           | 14                          |
| 2       | 61.4                   | 0.12                   | 15.9      | 15.3           | 16.9                        |
| 3       | 92.1                   | 0.23                   | 17.2      | 16.2           | 15.8                        |
| 4       | 122.8                  | 0.35                   | 18.1      | 19             | 17.5                        |
| 5       | 153.5                  | 0.48                   | 18        | 18.3           | 19.1                        |

CFD Rigid body*: this method used to find the swirl angle depend on swirl meter rotation (swirl meter setup as body), its available directly by monitor the change in Euler angle with time step in CFX solver.

CFD tangential velocity*: the details of this method in section 3 (equation 2)

Figure 6. Swirl angle comparison

Fig. 7 shows tangential velocity profile for case 1 and case 3. When analyzing these cases we find that minimum value for average velocity at 0.25d and 0.75d because of no slip condition while the maximum value in the center of swirl meter at 0.5d. Also the higher expected fluctuation for tangential velocity indicates good agreement with high swirl angle results as obtained.
From Fig. 8 we can see reduced pressure concentrated near side wall region which means water particles concentrated at that point where pressure is less or vortex is created at low pressure region due to water particles flows towards less pressure area with increased velocity, from same figure we can see the helical path of streamlines for vortex formation. This can be also seen with the help of velocity stream lines which are shown in Fig. 10. Fig. 9 shows the tangential velocity plane for free surface vortex which is inversely proportional to vortex radius.

![Figure 7. Tangential velocity profile (Case 1 & Case 3)](image)

**Figure 7.** Tangential velocity profile (Case 1 & Case 3)

![Figure 8. Free surface vortex, helical path and pressure distribution](image)

**Figure 8.** Free surface vortex, helical path and pressure distribution
By the experimental test and CFD the four type of free vortex have been observed as shown in Fig. 11 and Fig.12. The way free surface vorticity was observed by CFD depends on specifying different air volume fraction for each type. It was discovered that 0.001-0.005 was a good range to observe free surface vortex. From Fig 11 type 1 is the first stage of vortex formation which was observed at 0.005 air volume fraction. The flow behaves as a constant swirl of surface and very small deformation of surface curvature Type 2 shows surface dimple and deformation and this type was observed at 0.002 air volume fraction value. In types 5 and 6 air entrainment starts to occur and pull air bubbles under the liquid surface and enters together with liquid via the hole up to the bell mouth. The minimum air volume fraction was observed at these cases. Generally the free surface vortex is unsteady behavior with various locations and duration. In present study for all cases the free surface vortices was near to sidewall 2 around bell mouth area and had a clockwise rotation. Fig. 13 shows the front view contour details for air volume fraction for type 5 at different time step.
Figure 11. free surface vortex types (CFD)

Figure 12. free surface vortex types (Experimental)

Figure 13. Air volume fraction
5. Conclusion
In our study, numerical simulation and experimental test was carried out to estimate the swirl angle and vortex prediction in a pump sump model. The following results were obtained:
1. The five cases studied showed good agreement for swirl angle analysis between CFD and experimental results.
2. CFD and experimental result show good prediction of the types and location of vortex.
3. The free surface vortex is unsteady behavior with changing location and duration of vortex. Tangential velocity for free surface vortex is inversely proportional to radius. The dynamic pressure is lowest in the core region, and increases as one moves away from it.
4. The differences in the results for swirl angle analysis can be due to some unavoidable errors in experiments such as the water flow causing the swirl meter to fluctuate and not completely turn. CFD results could not predict the strength of vortices because the diameters of vortices are usually much smaller compared with the computing grid size.

6. References
[1] Choi J.W. (2012). A Study on the Flow characteristics of the Sump Pump Station with the AVD Installation and the Performance Analysis of the Mixed Flow Pump.(Ph.D. dissertation).Korea Maritime and Ocean University.
[2] Choi J W, Park N S, Kim S S, Park S S and Lee Y H,2012, "Study on Performance Analysis of Pump within Sump Model with AVD installation by CFD” Journal of Korean Society of Water and Wastewater, Vol 26,No3,pp463-469.(In Korean)
[3] Chang-G K, B H Kim, Y H Lee. (2015).Experimental and CFD analysis for prediction of vortex and swirl angle in the pump sump station model. International Symposium of Cavitation and Multiphase Flow (ISCM 2014).
[4] ANSI/HI 9.8,”American National Standard for Rotodynamic pumps for pump intake design”. (2012).Parsippany, NJ pp 7-40.
[5] Glīch, JohannF. Centrifugal Pumps".1sded Berlin: Springer-Verlag, 2010.pp16-20
[6] Luca C ,Matteo N,Gianfranc 2013 Sep. “Experimental Study on Unstable Vortices and gas Entrainment” Sci. Technol. 52,221-229
[7] Choi J W,Choi Y D, Kim C G and Lee Y H,2010,Flow uniformity in a multi-intake pump sump model Journal of Mechanical Science and Technology 24(7)1389-1400.
[8] Chang-Gu Kim. (2013). A study on flow Analysis for Pump Sump Model and Design Methods of Anti-Vortex Device.(Ph.D. dissertation).Korea Maritime and Ocean University.
[9] Iwano R, Shibata T,Nagahara T and Okamura T,2002,”Numerical Prediction method of asubmerged vortex and its application to the flow in Pump Sumps with and without a Baffle Plate Proc.of the 9th Int. Sump.on Transport Phenomena and Dynamic of rotating Machinery(Honolulu,Hawai,USA,10-14 Feb 2002)pp1-6
[10] Ansys inc,2013, Two equation turbulence models, Ansys CFX-Solver Theory guide.