An Investigation Of Surface Vortices Behavior In Pump Sump

Won-Tae Kang[1], Byeong Rog Shin[2*]

A numerical investigation on a suction vortices, free vortices and subsurface vortices behavior in the model sump system with multi-intakes is performed. A test model sump and piping system were designed based on Froude similitude for the prototype of the recommended structure layout by HI-9.8 Standard for Pump Intake Design of the Hydraulic Institute. A numerical analysis of three dimensional multiphase flows through the model sump is performed by using the finite volume method of the CFX code with multi-block structured grid systems. A k-ω ShearStressTransportturbulencemodelandthe Rayleigh-Plesset cavitation model are used for solving turbulence cavitating flow. From the numerical analysis, several types of vortices are reproduced and their formation, growing, shedding and detailed vortex structures are investigated. To reduce abnormal vortices, an anti-vortex device is considered and its effect is investigated and discussed.

Keywords: Pump Sump, Vortex Cavitation, Free Surface Vortices, Subsurface Vortices, Swirl Angle, Anti-vortex Devices, CFD

1. Introduction

Pump system is one of the most important fluid machine system which delivers the energy to fluid by impeller. This system is used commonly in the industry because it has simple structure and covers a wide range of discharge flow rates and heads. However, during pump operation, cavitation, flow separation, pressure loss, vibration and noise occur often by flow unsteadiness and abnormality. Especially, air-entrained free and subsurface vortices observed in sump pumps seriously damage to pump system. According to the HI standard of Hydraulic Institute HI[1] or JSME criteria for a pump sump design[2], therefore, these vortices should be prevented and their disappearance must be verified by sump pump model test in the construction of pump station.

To reduce these vortices and for the advanced pump sump design with high performance, it is very important to know the detailed flow information in sump system. For this purpose, to date many researchers have made experimental and numerical studies on the flow in pump sump. However, due to the high cost for design and physical model test CFD analysis has recently considered as an effective tool to evaluate the flow around the suction intake in pumps. Iwano et al.[3] made a trial of an application of a numerical prediction method of a submerged vortex to the flow in pump sumps in order to increase the vortex resolution by the conventional code based on the Reynolds Averaged Navier-Stokes equations. Detailed vortex flow phenomena including vortex cavitation, submerged vortex, vortex breakdown and vortex filament were investigated at the flow simulation around bell mouth with and without baffle plate in the single intake. Regarding the CFD prediction and model experiment on suction vortices in pump sump, recently Okamura et al.[4] performed a benchmark test by using several CFD commercial codes and reviewed the results to check their applicability to the design of the pump station instead of the expensive conventional experimental method. Kim et al.[5] studied about the characteristics of the subsurface vortex in the three-different pump sump and showed the usefulness of CFD to predict the subsurface vortex generation. Like this, many parametric studies to increase hydraulic performance of pump intakes have made as reviewed so far. Unfortunately, however, detailed behavior of free surface and subsurface vortices, minimum water levels incepting the vortices, swirling angle and so on are not investigated yet.

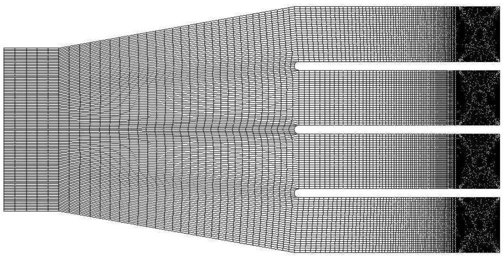
1 Student Member, Graduate School, Changwon National University
2 Life Member, Dept. Mechanical Engng., Changwon National University. : 055) 213-3609
* Corresponding author E-mail: brshin@changwon.ac.kr

In this paper, a numerical and investigation on a free and subsurface vortices behavior in the model sump system with multi-intakes is performed. Several types of free surface and submerged vortex are reproduced by the numerical analysis. To reduce these abnormal vortices, anti-vortex devices (AVD) are considered and its effectiveness is also investigated.

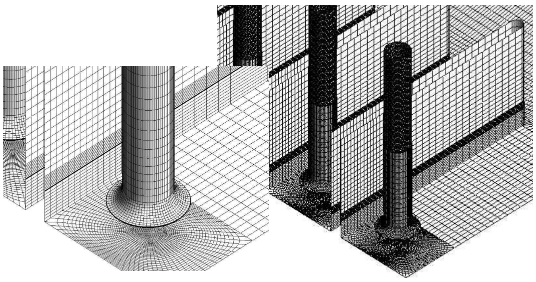
2. Numerical Simulation

To check the occurrence of visible surface vortices, and to investigate detailed structure and behavior of them a numerical simulation of three dimensional multiphase flows in the model sump was performed by using the finite volume method of the CFX code [6]. A multi-block structured grid system with about 1x106grid points was applied as seen in Fig.1. The flow rate is Q=126.9 m3/s and 4-bell mouth operation. Specified flow rates are imposed on the up- and down stream boundaries and slip and non-slip wall boundary condition are given on the free surface and walls, respectively. The working liquid is the city waterat 20°C.

Fundamental equations are the continuity equation and the Reynolds Averaged Navier-Stokes equations. A k-ω Shear Stress Transport turbulence model[7] and the Rayleigh-Plesset cavitation model [8] were used for solving turbulence cavitating flow.



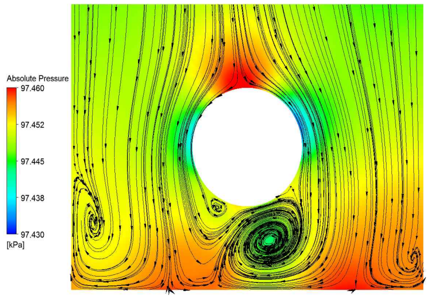
(a) Bottom view of model sump



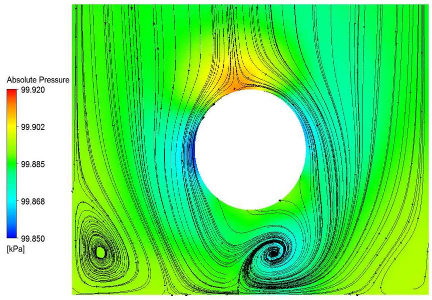
(b) Near bell mouth (c) Near intakes

Fig. 1 Computational Grid

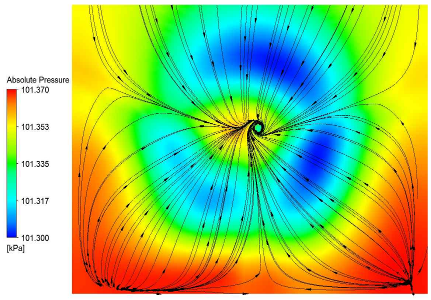
Figure 2 shows numerical results of streamlines and pressure distributions at several cross sections around the #1 bell mouth and suction pipe without the AVD. Here *d* represents inlet diameter of the suction pipe. A couple of surface vortices formed behind the suction pipe and both corners on the free surface are growing and shedding along the pipe. Then they are converged under the bell mouth and flowing to downstream. Detailed vortex structure, flow separation and pressure distributions were well simulated.



(a) On the free surface



(b) At 1*d* downstream section from the bottom



(c) Near the bottom wall

Fig. 2 Computational Result of Streamlines and Pressure Distributions

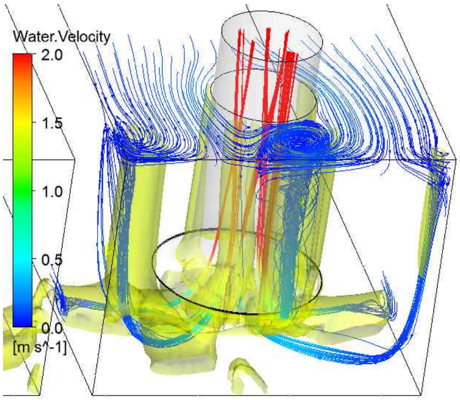
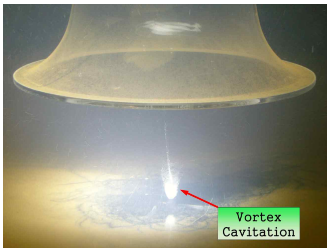


Fig. 3 Streams and Vortex Iso-surface Near #1 Bell Mouth and Suction Pipe

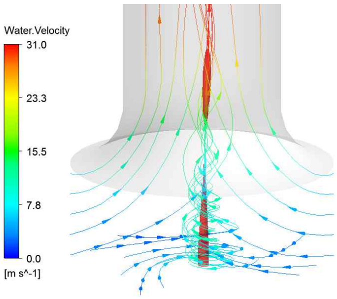
As illustrated in Fig.3, free surface vortices are merged into a strong vortex under the bell mouth and discharged with a high velocity. Relatively weak subsurface vortices also occurred on the both side walls but still no pulling air and no cavitation were shown in this case. A whole aspect of vortex flow behavior is well understood in this figure. Similar flow pattern was appeared for the #2 intake even though it is not shown here. But it was somewhat weak vortex flow because the flow passage is located near the center of the sump. The flow around the #3 and the #4 intake was almost the same as that of the symmetry intakes of the #2 and the #1.

In order to investigate the effect of the AVD computations was performed with and without the AVD, which was installed by setting a floor splitter under the bell as a purpose to reduce the swirl angle, flow separation and abnormal subsurface vortices. The optimal shape of splitter with a trapezoidal section was selected through a parametric study.

Figures 4 and 5 show computational results of streamlines near the bell in the case of sump with and without AVD. It can be seen that a strong vortex flow formed on the bottom under the bell is passing out through the intake in Fig. 4. Submerged vortex cavitation represented by iso-surface in red also occurred under the bell mouth in the low pressure region. Computational prediction in Fig. 4(b) agrees well with experimental observation of Fig. 4(a). Inside of the intake pipe the vortex cavitation is again developed due to the increased velocity in Fig. 4(b). On the other hand, the subsurface vortex was disappeared by just installing the AVD as shown in Fig. 5. The suction flow is almost straight and the swirl is evidently weak.

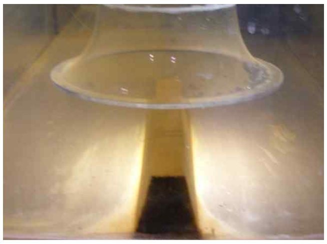


(a) Observed

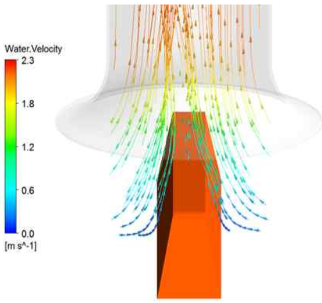


(b) Calculated

Fig. 4. Comparison of Computed Vortex Cavitation with Experiment (without AVD)

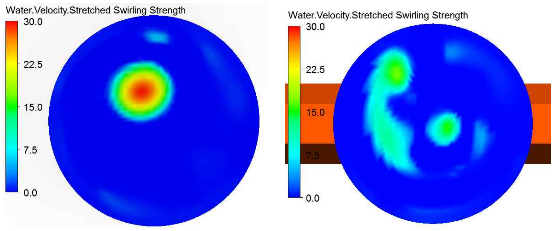


(a) Observed



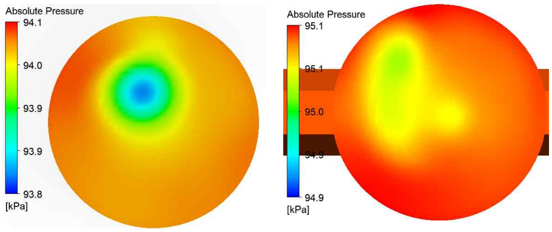
(b) Calculated

Fig. 5. Flow Pattern Obtained by Computation and Experiment (with AVD)



(a) without AVD (b) with AVD

Fig. 6. Swirling Strength Distribution



(a) without AVD (b) with AVD

Fig.7 Pressure Distribution

Figures 6 and 7 show computational results of swirling strength and pressure distributions on a cross sectional plane of intake pipe at 3.3d downstream from the bottom of the bell. The swirling strength at the center of submerged vortex before installation of AVD is much stronger than that with AVD. On the contrary, pressure at the vortex center without AVD is lower than that with AVD. As investigated so far, installation of AVD is very effective to reduce the swirl and subsurface vortices.

3. Conclusion

A numerical investigation on a suction vortices behavior including cavitation in the model sump system with multi-intakes were performed. Through the multiphase flow analysis by CFD, free and subsurface vortices are reproduced and their formation, growing, shedding and detailed vortex structures were well investigated, so that it is very easy to understand the complicated vortex flow behavior. The installation of anti-vortex devices is very effective to reduce the swirl and subsurface vortices including vortex cavitation.

Acknowledgments

This work was supported by the National Research Foundation of Korea (NRF) grant No.2009-0083510 funded by the Korean government (MEST) through Multi-phenomena CFD Engineering Research Center and Underwater Vehicle Research Center (UVRC), ADD of Korea.

References

[1] 1998, Hydraulic Institute Standard, American National Standard for Pump Intakes Design, ANSI/HI 9.8.

[2] 1984, JSME, Standard Method for Model Testing the Performance of a Pump Sump, JSME S 004.

[3] 2001, Iwano, R., et al., "Numerical Prediction Method of a Submerged Vortex and Its Application to the Flow in Pump Sumps with and without a Baffle Plate," Proc. of 9th Int'l Symp. on Transport Phenomena and Dynamics of Rotating Machinery, pp.1-6.

[4] 2007, Okamura, T., et al., "CFD Prediction and Model Experiment of Suction Vortices in Pump Sump," Proc. of 9th Asian Int'l Conf. on Fluid Machinery, AICFM9-053.

[5] 2005, Kim, J.Y., et al., "Numerical Analysis of the Subsurface Vortices in the Pump Sump Models," Proc. of the KFMA Annual Meeting 2005, pp.593-597.

[6] 2007, ANSYS, CFX User's Manual, Version 11.

[7] 1949, Plesset, M.S., "The Dynamics of Cavitation Bubbles," ASME J. Appl. Mech., Vol.16, pp.228-231.