A Numerical Investigation of Water Level Effect on the Vortex in Intake Sump

B. SHIN[a)]

Abstract

The effect of the water level on free surface vortices in intake sump was investigated by numerical approach. A finite volume method with a RANS turbulence model and a VOF multiphase model was applied to solve the sump flow with a single intake channel. Multi-blocked structured grids and the open channel model were used to capture the flow behavior and their mutual interactions between air and water phases with higher accuracy. Air-water interface level, air-entrained vortex length, and air volume fraction contours and iso-surfaces were used to visually identify the location and shape of the free surface as well as surface vortices. By monitoring the vortex length with varying water level, it was found that that lower water levels induced stronger free surface vortices. The behavior of surface vortices and air bubbles caused by breaking off air-entrained vortices was explained well by using the volume rendering method and air volume fraction contours. The predicted velocity distributions at the entrance of bell mouth were in a good agreement with those of experiments.

Keywords: Numerical analysis, VOF multiphase model, Free surface vortex, Vortex length, Intake sump

1. INTRODUCTION

One of the most important systems in fluid engineering is the pump system with the intake devices designed in accordance with their purpose. The intake devices in sumps play specially an important role to keep stability and uniformity of flow before entering into the pump for improvement of efficiency of pump system. Therefore, the applications of intake devices such as a sump type with better models need to be implemented as far as possible. During operating pump system, the unexpected phenomena such as cavitation, head and pressure loss, vibration and noise need to be considered especially in which air-entrained free and submerged vortices appearing in sump pumps will damage seriously to pump system.

Therefore, in order to make an advanced design of the pump sump preventing the occurrence of undesirable vortices, a great deal of research to get detailed flow information in the sump has been carried out. For instance, Rajendran and Patel[1)] made detailed measurements of vortices in a physical model pump-intake bay using particle image velocimetry (PIV) to improve the understanding of vortices formed in the bay. The experiments obtained the quantitative information such as the number, location and size of the vortices. In recent years, Kawakita, et al.[2)] constructed three scaled model sumps with different diameters of the suction bell, and investigated the similarity law in the flow of these scaled sumps through the model test. Comparing the occurrence limits of air entraining vortex and submerged vortex, a kinetic similarity was obtained only at the same velocity condition. Such experimental works, however, involve many limitations on the modification of the geometry and the variation of test options, and require a lot of manufacturing time. In addition, the similarity in the model testing is not clear yet.

A CFD benchmark test was performed by Matsui, et al.[3)] to examine the characteristics of various CFD codes and to make a comparative analysis of the prediction accuracy for the flow in a simplified sump configuration. Single phase flows in the sump with a boundary condition of a fixed slip wall without friction for the free surface was computed, and predictions such as local velocity distributions, the intensity of vortices and flow patterns were then compared with experimental data. By applying the CFD codes and turbulence models, the predictions were somewhat different in the distribution of vorticity, but they showed that some CFD codes could predict the visible vortex occurrence and its location with enough accuracy for industrial use. Recently, Kang, et al.[4)] carried out a numerical investigation of surface vortices in a realistic model sump with a multi-intake by using the CFX. A k-ω Shear Stress Transport (SST) turbulence model and a cavitation model were used. By the turbulence cavitating flow analysis, the surface vortices and their structures, formation, growing and discharge were well simulated. As can be seen above, there is a trend toward CFD simulation for solving fluid flow problems, and CFD is being regarded as a suitable alternative for evaluating sump performance and has contributed to improving the flow condition for the optimum design

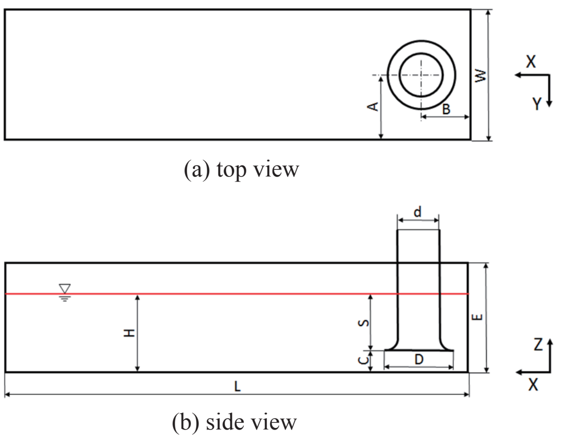
a) Professor, Dept. of Mechanical Design Systems Engineering

of pump sumps. However, since most simulations treat the free surface flow as a single phase flow with a flat free surface or fixed slip wall without friction, the behavior of free surface vortices has not been satisfactorily clarified yet. The computation of the sump flow, therefore, has to give attention to dealing with the free surface because the free surface flow has significant influence on the occurrence of surface vortices in a real sump construction. For the accurate prediction of gas-liquid multiphase flows, the solution method of the multiphase flow is also required.

In this paper, a numerical simulation of 3-D free surface flows in pump suction sump is performed to investigate the behavior of surface vortices with respect to the water level. A RANS model with the *k-ω* SST turbulence model is solved by finite volume method. A volume of fluid (VOF) multiphase model and an open channel model considering the effect of friction and surface tension are applied to analyze the free surface flow problem with accurate. By using the volume rendering and iso-surface contour of air-water interfaces, velocity profiles and pressure distributions, the behavior of the free surface and surface vortices at different water levels is investigated and discussed.

2. NUMERICAL METHOD AND BOUNDARY CONDITION

The computational flow model with a single pump-intake is the same as that used in the previous work[5] as shown in Fig.1. In this model, the inlet bell mouth diameter D was used as the basic design parameter. The intake pipe with the inside diameter d of $0.6D$ is located in the middle of intake channel width W ($=2D$) at a distance B of $0.75D$ from the back wall to the pump inlet bell center. The thickness of the intake pipe and bell is set to $0.04D$. The bottom clearance C from floor is equal to $0.4D$. The length L of rectangular intake channel is $7.32D$. The origin of coordinates was located at the center of the bell mouth on the floor.



(a) top view

(b) side view

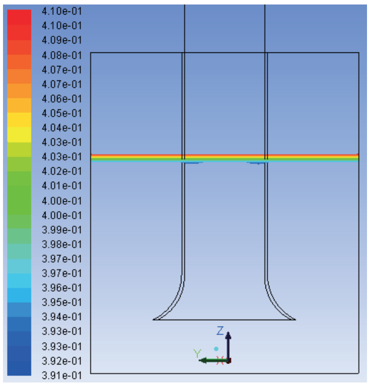
Fig. 1. Schematic illustration of computational domain.

Numerical simulation of 3-D multiphase flow in a pump intake sump was performed by solving the RANS equations with *k-ω* SST turbulence model discretized by finite volume method as done in the previous work[5]. The VOF multiphase model and an open channel model were applied to analyze the free surface flow problem with accurate. Furthermore, the body force by the gravitation and the volumetric tension force to consider the effects of surface tension at fluid-fluid interface were introduced.

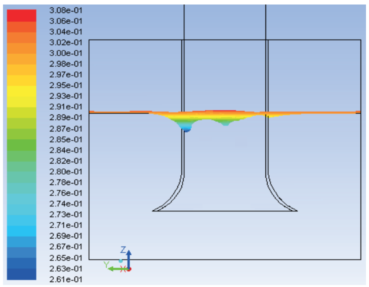
For the boundary conditions, the Dirichlet boundary condition of pressure and mass flow rate were imposed on the inlet and outlet boundary respectively with the flow rates. On the boundary between the air and water interface, an open channel model combined with the VOF model in the FLUENT[6] was applied to accurately capture the air-water interface and flow interactions between gas and liquid phases. A multi-block structured computational grid with 851,000 hexahedral elements was used. This grid was generated with high density at the free surface region and near the bell mouth as well as intake pipe to capture the sensitive behaviors of the fluid-fluid interface. The grid dependency and convergence were also confirmed. Details for the numerical method, boundary conditions and computational grid will be referred to the previous paper[5].

3. NUMERICAL RESULTS

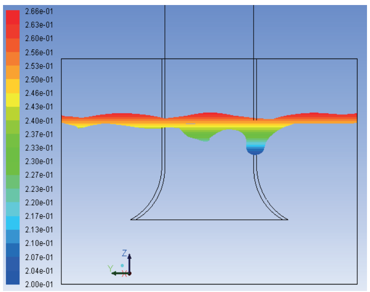
Figure 2 shows a computational result of an instantaneous elevation of air-water interface near intake pipe at different water levels and a fixed flow rate of $190.35 m^3/h$. It was captured in the front side of the channel. The position of free surface and free surface vortices were specified by the z-coordinate, so that the scale, distribution and overall shape of free surface vortices and the wavy pattern of the interface are easy to visually identify and understand. In this investigation, the scale of free surface vortices and their length were inversely proportional to the water level. At high water level of $H=1.6D$ in Fig.2(a), a very calm and shallow free surface vortex resembling a surface swirl[7] appeared relatively steadily behind the suction pipe. When the water level was gradually decreased, however, the appearance of free surface vortex and its scale were increased accordingly. At water level of $H=1.2D$ in Fig.2(b), a pair of almost symmetric vortices occurred in the early stage of computation, and then gradually changed into the asymmetric pattern mainly due to the flow instability and the surface wave derived from the water intake structures with low water level[5),7]. The larger vortex on the left hand side seems to be strong but it could not induce the dye core and pulling air bubbles. In the case of low water level of $H=1.0D$ in Fig.2(c), significantly large surface vortices appeared and then they showed very dynamic behaviors including the appearance of pulling air bubble as investigated in detail in the previous work.



(a) H=1.6D



(b) H=1.2D



(c) H=1.0D

Fig. 2. Instantaneous elevation of free surface and free surface vortices at different water levels and Q=190.35m^3/h.

Figure 3 shows a time averaged vortex length with different water levels. It is easy to see that the vortex length and its slope decreased with the water level, while it was increased with the flow rate.

Figure 4 shows predicted velocity profiles along the y direction at the bell mouth center and z of 0.38D from the sump floor (just below the bell mouth) at the flow rate of 237.94 m^3/h. u, v and w represent the velocity components in the x, y, and z direction respectively. V_m denotes the mean velocity of the bell mouth for given flow rate. It can be observed that with the water level the amplitude of velocity fluctuation is decreased, and from the change of the velocity direction of u, v the existence of a pair of vortices and their strength is figured out. In the vicinity of the bell mouth, axial velocity, w was predominant compared with the other velocities and formed a small velocity valley between the two vortices as well. The overall tendency of the predicted velocity distributions qualitatively agreed well with the PIV measurements[8].

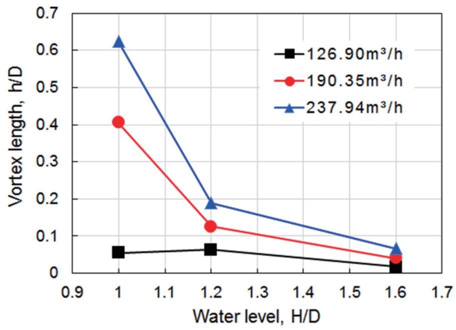


Fig. 3. Time averaged vortex length.

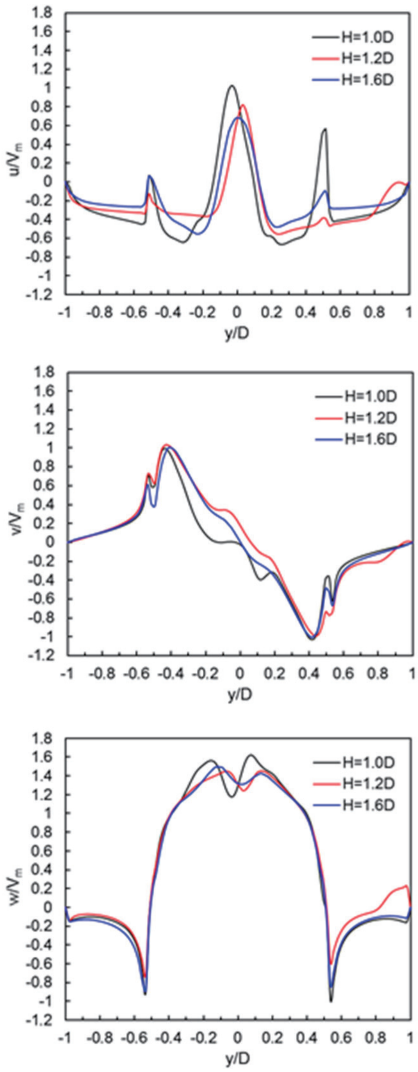
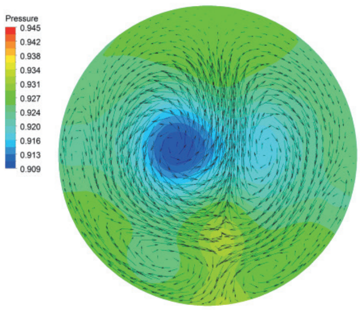
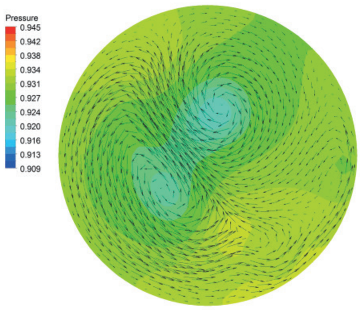


Fig. 4. Instantaneous velocity u, v and w profiles along the y direction at bell mouth center and Q=237.94 m^3/h.

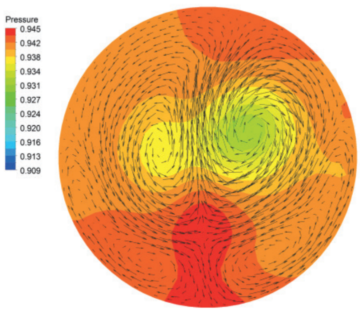
Figure 5 shows the computational results of instantaneous velocity vectors and pressure contours at z =1.0D cross section of intake pipe at flow rate of 237.94 m^3/h. It can be seen that the intake flow entered through the bell mouth is discharged accompanying secondary flows with several vortices, and the counter rotating pair vortices caused by the free surface vortex are significantly stronger than the others. In general, the pressure is increased in proportion to the water level due to the hydrostatic relations and relatively high at impinging area of velocity. In addition, it indicated that lower water level made stronger vortices with low pressure.



(a) H=1.0D



(b) H=1.2D



(c) H=1.6D

Fig. 5. Instantaneous velocity vectors and pressure contours at z=1.0D and Q=237.94 m^3/h.

4. CONCLUSION

A numerical study of 3-D free surface vortices occurred around the intake pipe in pump sump was carried out by using finite volume method for RANS equations and k-ω SST turbulence model. A multi-block structured grid and a VOF multiphase model as well as the open channel model were used to accurately solve the multiphase flow problem in the pump sump. A single intake channel flow with variation of water level was simulated and investigated. Conclusions are summarized as follows.

From the investigation of the location of air-water interface and the air-entrained surface vortex length, the variation of form and shape and the behavior of free surface vortex corresponding to each cases of given flow rates and water levels were clarified. By using the volume rendering method and air volume fraction contours, the formation process of pulling air bubbles and the structure of free surface vortices became clear. By monitoring the air-water interface with variation of water levels, it was confirmed that the water level has a significant effect on the occurrence of free surface vortex. The lower water levels induced higher surface vortex generation. The overall structure and strength of swirling flow caused by the free surface vortices are known through the predicted velocity and pressure distributions at the entrance of bell mouth, and the predicted velocity distributions qualitatively agreed well with experiments.

REFERENCES

1) Rajendran, V. and Patel, V., Measurement of Vortices in Model Pump-Intake Bay by PIV, J. Hydraul. Eng., Vol.126 (2000), pp. 322-334.
2) Kawakita, K., Matsui, J. and Isoda, H., Experimental Study on the Similarity of Flow in Pump Sump Models, Proc. 26th IAHR Symp. on Hydraul. Machinery and Systems, Beijing, 2012.
3) Matsui J., Kamemoto, K. and Okamura T., CFD Benchmark and a Model Experiment on the Flow in a Pump Sump, Proc. 23rd IAHR Symp. on Hydraul. Machinery and Systems, Yokohama, 2006.
4) Kang, W.T., et al., An Investigations of Cavitation and Suction Vortices Behavior in Pump Sump, Proc. ASME-JSME-KSME Joint Fluids Eng. Conf., Hamamatsu, 2012.
5) Shin, B., A Numerical Investigation of Flow Rate Effect on Vortices in Intake Sump, Memoirs of Faculty of Eng., Univ. of Miyazaki, N.47, 2018, submitted.
6) ANSYS group, FLUENT User's Manual, Ver. 13, 2010.
7) Claxton, J., et al., American National Standard for Pump Intakes Design, Hydraulic Institute, ANSI/HI 9.8, 1998.
8) Okamura, T., Kamemoto, K. and Matsui, J., CFD Prediction and Model Experiment on Suction Vortices in Pump Sump, Proc. 9th Asian Int. Conf. on Fluid Machinery, Jeju, 2007, AICFM9-053.