CFD study on flow characteristics of pump sump and performance analysis of the mixed flow pump

Y X Zhao¹, C G Kim¹ and Y H Lee²

¹Department of Mechanical Engineering, Graduate School, Korea Maritime University, Busan, 606-791, Korea
²Division of Mechanical Engineering and Information, Korea Maritime University, Busan, 606-791, Korea

E-mail: lyh@hhu.ac.kr

Abstract. Head-capacity curves provided by the pump manufacturer are obtained on the condition of no vortices flowing into the pump intake. The efficiency and performance of pumping stations depend not only on the performance of the selected pumps but also on the proper design of the intake sumps. A faulty design of pump sump can lead to the occurrence of swirl and vortices, which reduce the pump performance. Therefore, sump model test is necessary in order to check the flow condition around intake structure. Numerical simulation is a good facility for reducing the time and cost involved throughout the design process. In this study, the commercial software ANSYS CFX-13.0 has been used for the CFD analysis of the pump sump. The effect of an anti-vortex device (AVD) for the submerged vortex has been examined. Hydraulic performances for the head rise, shaft power, pump efficiencies versus flow rate are studied by the performance curves. In addition, numerical simulation of cavitation phenomenon in a mixed flow pump has been performed by calculating the full cavitation model with k-ε turbulence model. According to the result, the efficacy of the AVD to ensure the uniform flow conditions around the pump intake is confirmed. From the numerical analysis, the inception of cavitation is observed on the suction surface where the leading edges meet the tip, and then the cavitation zone expands.

1. Introduction

Sump is designed to supply uniform swirl free flow to pump intake and satisfy the conditions of the hydraulic design of the pump. Undesirable flow conditions induce cavitation, additional vibration and excessive blade loads, which directly affect the running efficiency and security performance of the pumping station. Various researches show the flow conditions around the sump intake structure are quite complicated. 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.

As an undesirable phenomena caused by the adverse flow condition, cavitation is the process of the formation of vapor bubbles within a liquid where flow dynamics cause the low pressure. With the rapid formation, growth and collapse of the bubbles, cavitation manifests in the form of pump
performance decrease, vibration, additional noise increase and even the equipment damage. 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. Moreover, there are many correlated studies [7]-[8].

In this paper, the numerical analysis of a mixed-flow pump sump model is conducted to predict the flow characteristics around the intake bell. Furthermore, the efficacy of an anti-vortex device (AVD) and cavitation phenomenon of the pump are analyzed.

2. Numerical method

The computational study is applied on a single-intake rectangular sump model. As illustrated in Fig 1(a), the total flow direction length of the model is 30m, with a long channel of 2.6m in width, 18m in length and a baffle about 8.6m distance from the intake bell center. The width of the intake channel is 4m, and the center of the intake bell is located at 1.8m, 2m and 0.7m from rear wall, side wall and bottom, respectively. ANSYS ICEM CFD is used for the grid generation. Fig 1(b) shows the mixed flow pump with hexa-hedral mesh. The mixed flow pump consists of 5 blades of the impeller, while 9 blades of the diffuser and the diameter of the impeller inlet is 1.096m. Fig 1(c) shows the mixed flow pump sump model with an AVD installed. The trident shaped AVD installed under the pump intake is made up of three wall fillets (800mm height) and one central splitter (200mm height).

The total node number generated for the model is over 5 million. The standard $k$-$\varepsilon$ turbulence model is adopted as the turbulence model. The inlet section at the entrance to the sump is specified as inlet boundary with the total pressure condition; meanwhile the mass flow rate is specified as outlet condition at the outlet section of the pump.

![Figure 1. Pump sump model.](image)

3. Results and discussion
3.1. Performance analysis of the mixed flow pump sump model
The design volume flow rate of the pump sump model is 21,700 m$^3$/hr. The performance analysis is conducted via flow rate changed from 50% to 140% of the design flow rate. The speed of the rotor is constant, i.e. 423 rpm.

![Figure 2. Performance curve for the mixed flow pump with the sump.](image)

Fig 2 shows the hydraulic performance of flow rate versus head, shaft power and pump efficiency of the pump sump model. The performance curves show that as the flow rate increases the head of pump decreases gradually; the shaft power and efficiency increase first and then decrease. The shaft power increases up to 1527.8kW at the flow rate of design. The best efficiency point (BEP) is at the design flow rate with the corresponding efficiency of 89.6%.

3.2. Flow characteristics analysis

![Figure 3. Streamline pattern around the intake structure.](image)

Fig 3 shows the comparison of streamline pattern around the pump intake structure with and without the AVD at the design flow rate. Near the side wall and the rear wall in the vicinity of the intake, vortex formation is observed and with the AVD installed the vortex is suppressed.
To check the vortex intensity around the intake structure quantitatively, Fig 4 shows the vorticity distribution comparison of the sump without and with the AVD installed both in the flow direction and the channel width direction. The horizontal axis coordinate $x=0.0$ means the center of the intake bell mouth, and $H$ is the distance of the cross sectional plan from the bottom wall, where $H=0.7m$ is the same height of the bell mouth inlet.

Fig 4 shows at the radius region of the bell mouth inlet wall, maximum value of vorticity is obviously higher than those at other heights, i.e. $H=0.7m$, $x=\pm 1m$, which means flow with high vorticity near the bell mouth tip. Fig 4 (a) shows the vorticity distribution in the flow direction at each height. Below the height of 0.7m, vorticity almost disappears except near the center of the bell mouth. Fig 4(b) also shows that below the height of 0.7m, there are only symmetrical vorticity values near the center of the bell mouth. In the comparison of the vorticity distribution with and without the AVD, though the vorticity of the region below the bell mouth is small, relatively lower values of the maximum vorticity are obtained by installation of the AVD. This implies that the possible vortex occurrence around the pump intake is reduced.

Figure 4. Comparison of the vorticity distribution.

3.3. Cavitation phenomenon analysis
Pump cavitation occurs when the local absolute pressure of the flow falls below the vapor pressure of the liquid. The net positive suction head (NPSH) is used to predict the cavitation performance, defined as:

\[ \text{NPSH} = \frac{P_0 - P_v}{\rho g} \]

where \( P_0 \) is the total pressure of the impeller inlet suction, \( P_v \) is the vapor pressure, \( g \) is the acceleration due to gravity and \( \rho \) is the density of the fluid.

**Figure 5.** Cavitation performance curves of the mixed flow pump.

Every pump has a critical cavitation spot, the required net positive suction head (NPSH\(_R\)), which is defined as the minimum necessary NPSH to avoid cavitation, usually a corresponding 3% or 5% head drop. Fig 5 shows the cavitation performance characteristics at different flow rate under the same rotational speed, where \( Q_0 \) is the design flow rate. When the inlet total pressure high enough, there is no cavitation and the head remains constant. When the inlet pressure decreases, the NPSH approaches the NPSH\(_R\), due to which, cavitation occurs and expands. Comparing the three curves, the NPSH\(_R\) for the flow rate 0.78\( Q_0 \), 1.00\( Q_0 \) and 1.41\( Q_0 \) is 11.4m, 7.2m and 11.0m, respectively, which means the cavitation performance of the \( Q_0 \) is the best. The curve shows that under the design working condition, i.e. the NPSH=13.4m, there is a considerable extent over the NPSH\(_R\)=7.2m, which means the pump at the BEP condition has met the requirement of the cavitation performance.

**Figure 6.** Vapor volume fraction distribution on blade surface.

To clarify cavitation phenomenon in flow passages of the mixed flow pump, the vapor volume fraction distribution at design flow rate condition is plotted in Fig 6.
Cavitation appears on the blade suction surface where the leading edge meets the tip, as NPSH decreases, cavitation zone expands. Although NPSH=13.4m is in the NPSH safety extent, there is slight cavitation only on the suction surface, whose influence to the pump performance is negligible. However, further reduction in the NPSH R=7.2m, like when NPSH=6.1m under the severe cavitation condition, the cavitation region takes about 50% area of the suction surface, i.e. serious flow passage blockage, which results in a major deterioration in the pump performance. Cavitation on the pressure surface shows the similar situation, the inception of cavitation is observed quite late due to the higher pressure distribution.

4. Conclusion
Based on the numerical calculation of the mixed flow pump sump, the efficiency of the sump pump, the effectiveness of an AVD and the cavitation phenomenon are predicted.

1. In the hydraulic performance analysis, the mixed flow pump with the sump shows high efficiency of 89.6% at the BEP condition. And it is confirmed that the pump operated under the BEP condition has met the requirement of cavitation performance.

2. With the AVD installed, the reduction of the maximum vorticity values at each height is verified, which decreases the submerged vortex intensity.

3. Cavitation first occurs on the blade suction surface near the leading edge and the shroud. As NPSH decreases , the cavitating region is spread out over the blade and the serious blockage in flow passage caused by cavitation results in a sharp reduction of the pump head.

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 Engineering Division Summer Meeting FEDSM (Washington, USA, 21–25 June 1998) 98 21-25
[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 February 2002) 1-6
[3] Choi J W, Choi Y D, Kim C G and Lee Y H 2010 Journal of Mechanical Science and Technology 24(7) 1389-1400
[4] Van Os M J, De woerd J O 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 Engineering Division Summer Meeting FEDSM (Vancouver, British Columbia, Canada 22-26 June 1997) pp 22-26
[5] Dupont P and Okamura T 2003 International Journal of Rotating Machinery 9(3) 163–170
[6] Li J, Liu L J, Li G J and Feng Z P 2007 Journal of Engineering Thermophysics 28(6) 948-950 (in Chinese)
[7] Chang S P and Wang Y S 2012 Journal of Drainage and Irrigation Machinery Engineering 30(2) 171-180 (in Chinese)
[8] Tamura Y, Fukaya M and Matsumoto Y 2009 Prediction of Impeller Speed Dependence of Cavitation Intensity in Centrifugal Pump Using Cavitating Flow Simulation with Bubble Flow Model Proc. of the 7th International Symposium on Cavitation (Ann Arbor, Michigan, USA, 16-22 August 2009)