Numerical investigation of cavitation in centrifugal pump impeller for prediction of NPSH

Jugnhyun Lee, Department of Mechanical Engineering, University of Sogang, Seoul, Korea
Nahmkeon Hur, Department of Mechanical Engineering, University of Sogang, Seoul, Korea
Insik Yoon, Dooch pump, Gyeonggi, Korea

ISROMAC
Since 1985
Long Abstract

1. Introduction

Cavitation is one of the hazardous fluid dynamic phenomena which frequently encountered in hydraulic machinery. Working under cavitation condition leads to serious damage of blades and case. Many experiments and numerical simulations have been investigated on this problem from various aspects.

Delgosha et al. [1] investigated two-dimensional curvature blade geometry in cavitating and noncavitating conditions using different experimental techniques. They also simulated a three-dimensional numerical model in order to study cavitating flows. Steinmann et al. [2] investigated the unsteady cavitating flow in a vortex pump. The CFD results turned out to be appropriate to increase the knowledge about vortex pumps but calculated values were not fully accurate by considering experimental data. Zhao et al. [3] simulated a cavitation flow on a 2D NACA0015 hydrofoil under high pressure and temperature. The heat absorption in the cavitation area reduces the local temperature and the saturated vapor pressure. ZANG and CHEN [4] analyzed the pump performance drop is relevant to the instability of cavitating flow on the blade suction surface. LIU et al. [5] found when the cavitation number decreases, the length of the sheet cavities grows substantially. Kim et al. [6] analyzed the cavitation phenomena occurring in the centrifugal pump, and flow characteristic inside the pump impeller caused by cavitation through a numerical analysis.

Computational fluid dynamics provides range of possibilities to predict the vaporization in the impeller region that can occur under different conditions. In this paper, the unsteady cavitating simulation was performed to calculate NPSH values. The sudden pressure drop in low NPSH condition is discussed.

2. Numerical simulations

The schematic view of the centrifugal pump model is shown in Fig. 1. The impeller is shroud type and the number of blades is five. The impeller inlet radius is 65mm and outlet radius is 138mm. For this simulation, design flow rate 25m^3/hr is applied to the inlet, and simulations were carried out at fixed impeller rotation speed 3000rpm. The Rayleigh-Plesset cavitation model and the VOF (Volume of Fraction) method are adopted to describe the cavitation phenomena. Furthermore, in order to increase the accuracy of simulations, the CFL (Courant Friedrichs Lewy) number was set up 0.1.

3. Results and Discussions

The simulations are carried out design flow rate (25m^3/hr) at a fixed rotational speed of 3000rpm. Figure 2 shows the total dynamic head drop curves with different NPSH values obtained form the CFD results. The total dynamic head is dramatically decreased near the NPSH 6m due to the interaction of rotating cavitation with the impeller. Because of low NPSH, the fluid on the suction side of the impeller is going to vaporize. Therefore, cavitation is observed in the impeller leading edge and trailing edge of the suction area.

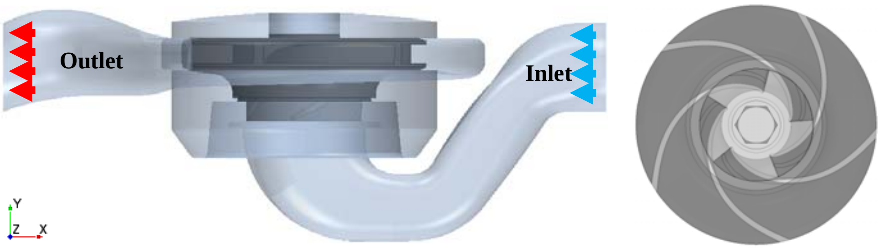

Figure 1. Schematic view of Centrifugal pump model

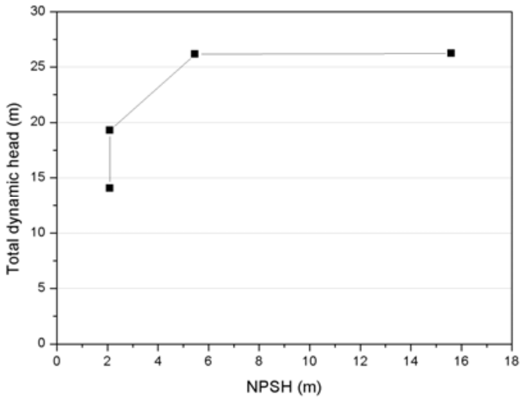

Figure 2. Total dynamic head versus NPSH in 3000rpm

4. Concluding remarks

The cavitating flows has been studied to predic the cavitation behavior in the centrifugal pump. Simulation results were presented and analyzed in various NPSH values. More simulation result need to investigate between NPSH 2~6m for accurated starting point of pressure drop. After that, the prediction of head breakdown can find for the critical NPSH. In the present paper shows that numerical simulation can provide useful information for the operating condition in centrifugal pumps.

References

[1] O. C. Delgosha, R. F. Patella and J. L. Rebound. Experimental and numerical studies in a centrifugal pump with two-dimensional curved blades in cavitating condition. *Journal of Fluids Engineering*, 125:970–978, 2003.

[2] A. Steinmann, H. Wurm and A. Otto Numerical and experimental investigations of the unsteady cavitating flow in a vortex pump. *Journal of Hydrodynamics*, 22:324–329, 2010.

[3] W. G. Zhao, L. X. Zhang and X. M. Shao Numerical simulation of cavitation flow under high pressure and temperature. *Journal of Hydrodynamics, Ser. B*, 23:289–294, 2011.

[4] R. ZHANG and H. X. CHEN Numerical analysis of cavitation within slanted axial-flow pump. *Journal of Hydrodynamics, Ser. B*, 25:663–672, 2013.

[5] H. L. LIU, D. X. LIU, W. Yong, X. F. WU, W. A. N. G. Jian and D. U. Hui Experimental investigation and numerical analysis of unsteady attached sheet cavitating flows in a centrifugal pump. *Journal of Hydrodynamics, Ser. B*, 25:370–378, 2013.

[6] M. J. Kim, H. B. Jin, C. H. Son and W.J.Chung. Numerical analysis on cavitation of Centrifugal pump. *Korean society for fluid machinery*, 16:27–34, 2013