A comparative study on accurate prediction of cavitation characteristics for head drop in a mixed-flow pump with steady- and unsteady-state analysis

Yong-In Kim¹,², Sung Kim², Hyeon-Mo Yang², Kyoung-Yong Lee², Yong-Kab Lee³, and Young-Seok Choi¹,²†

¹ Green Process & Energy System Engineering, University of Science & Technology, Daejeon, Republic of Korea
² Thermal & Fluid System R&D Group, Korea Institute of Industrial Technology, Cheonan, Republic of Korea
³ Anflux Co., Ltd., Seoul, Republic of Korea

E-mail: yschoi@kitech.re.kr

Abstract. The cavitation characteristics for the head drop in a mixed-flow pump were investigated with the steady- and unsteady-state analysis. Two mixed-flow pump models exhibiting a different incidence angle were analyzed under the cavitation condition. The model with a larger incidence angle obtains relatively poor suction performance. Moreover, the steady- and unsteady-state analysis indicate different head level under the cavitation condition. The results of the unsteady-state analysis are more accurate to the experimental results. The head of a fully convergent steady-state analysis is not even distributed in the head fluctuation range of the unsteady-state analysis. However, the head drop due to the decrease of inlet pressure has almost the same distribution and gradient. In addition, the amount and shape of the bubbles were presented with the time variation, which were also compared with the averaged result of steady-state analysis. The bubbles show a larger oscillation in the model which has a larger incidence angle. The oscillation of the bubbles is related to the magnitude of the head fluctuation.

1. Introduction
The cavitation of a pump develops and spreads due to local pressure drop at the impeller leading edge (LE) of the shroud span, causing the head drops, vibrations, and noise [1, 2]. In particular, the head drop in cavitation is a very important aspect in terms of energy saving, so that, more detailed analyses for these phenomena are required persistently. For the head drop characteristics in cavitation, pumps with faster head drop will have poorer cavitation characteristics than the pumps with a slower head drop. The head may fall faster or slower depending on the flow rate or design parameters of a pump [3-6]. In this study, cavitation performance with the specific design variable was analyzed at the design flow rate.

One of the most important design variables that determine the cavitation performance of a mixed-flow pump is the incidence angle at the impeller inlet. The distribution of incidence angle has a direct effect on the local pressure distribution at the impeller LE [7]. In this study, two models with different incidence angles were studied with adjustment of the impeller inlet area and the blade angle. In addition, the results of both steady- and unsteady-state numerical analyses were compared with the tendency of head drops and shape of bubbles.
2. Mixed-flow pump models

Two models with the different incidence angles were designed under adjustment of the impeller inlet area and the blade angle. The inlet area is a design variable that determines the flow angle with the meridional component of the absolute velocity. The flow angle can be also calculated directly with the difference between the blade angle and incidence angle as shown in Figure 1. The symbols $U$, $C$, $W$, and $\beta$ represent the rotational velocity component, absolute velocity component, relative velocity component, and blade angle, respectively. The subscripts $m$ and 1 indicate the meridional component and impeller inlet, respectively. Among the two models, the model with the smaller incidence angle was named as type A, and the last one with the larger incidence angle was named as type B. Figure 2 shows the meridional plane of the type A and B. The inlet radius ratio which is defined as the hub radius divided by the shroud radius ($r_{1h}/r_{1s}$) is 0.253 and 0.262 for A and B, respectively. Table 1 lists the solidity, the chord length divided by the trailing edge (TE) pitch ($C_l/S_2$), for each model.

Except for the inlet area and blade angle, all design variables and specifications were unaltered. The head coefficient ($\psi$) and the flow coefficient ($\phi$) calculated in equations (1) and (2) are 0.51 and 0.19. The rotational speed is 2400 rpm and the number of blades is five. The specific speed ($N_s$) which is defined as equation (3) is 2.43. The tip clearance and diffuser of the mixed-flow pump models were not considered.

\[
\psi = \frac{2gH}{u_2^2} \tag{1}
\]
\[
\phi = \frac{C_{m2}}{u_2} \tag{2}
\]
\[
N_s = \frac{\omega \sqrt{Q}}{(\beta H)^{\frac{3}{2}}} \tag{3}
\]

where $g$, $H$, $C_{m2}$, $u_2$, $\omega$, and $Q$ denote the acceleration due to gravity, total head, meridional component of the absolute velocity at the impeller outlet, tangential component of the rotational velocity at the impeller outlet, angular velocity, and volumetric flow rate, respectively.

![Velocity triangle and incidence angle at impeller inlet for each model](image)

**Figure 1.** Velocity triangle and incidence angle at impeller inlet for each model.
### Table 1. Solidity for each model.

| Span     | type A | type B |
|----------|--------|--------|
| Hub      | 1.7809 | 1.6691 |
| Mid      | 1.8448 | 1.6565 |
| Shroud   | 1.8710 | 1.6107 |

### Table 2. Results of the grid test.

| Total number of nodes | Head coefficient (\(\psi\)) [-] | Total efficiency [%] |
|-----------------------|----------------------------------|----------------------|
| 82,165                | 0.543                            | 92.9                 |
| 180,763               | 0.571                            | 93.1                 |
| 345,093               | 0.577                            | 93.7                 |
| 509,423               | **0.579**                        | **93.8**             |
| 1023,672              | 0.579                            | 93.9                 |

![Figure 2. Meridional plane of each model.](image)

3. **Numerical analysis methodology**

The shear stress transport (SST) standard turbulence model was applied with the commercial code, ANSYS CFX 16.1. The Rayleigh-Plesset cavitation model was selected to simulate the two-phase transition [8-10]. The computational domain consists of a rotating domain (impeller) and a fixed domain (inlet and outlet), and each boundary condition of the interface was given as the frozen-rotor method for steady-state analysis, and the transient rotor-stator method for unsteady-state analysis, respectively. The unsteady source term was given for every 3 degrees with seven revolutions. For the inlet and outlet, the pressure and mass flow rate conditions were applied, respectively. In cavitation analysis, head drop characteristics were obtained adjusting the boundary condition against the inlet pressure. To obtain the detailed flow phenomena in the pump, a full-passage numerical analysis was conducted for both type A and B.

The hexahedral grid system was constructed as shown in Figure 3. The grid systems near the wall boundary surface, impeller LE, and TE of the computational domain are shown as enlarged scale. The \(y^+\) value was set to less than 15 under the automatic wall function in ANSYS CFX 16.1, which can obtain appropriate numerical results [11, 12]. Table 2 lists the results of the grid test. As the total number of nodes increases, the numerical results for head coefficient and total efficiency are changed, and the values are almost constant from about 510,000 number of nodes. Therefore, the grid system was applied with the topology corresponding to about 510,000 nodes.
4. Results and discussion
The numerical results of the steady- and unsteady-state analysis for both models were validated with experimental data under ISO 5198 and ANSI/HI 1.6 standards [6]. Figures 4(a) and (b) show the comparison of type A and B, respectively. In terms of the inlet pressure, the cavitation coefficient ($\sigma$) is expressed as equation (4).

$$\sigma = \frac{2(P_1 - P_v)}{\rho u z^2}$$ (4)

where $\rho$, $P_1$, and $P_v$ denote the water density, inlet total pressure, and saturation vapor pressure, respectively.

Both models show that the head drop occurs as the inlet pressure decreases, and each gradient of the head drop is very similar between the experimental data and the numerical results. Each experimental data in Figures 4(a) and (b) are located at the lowest position because of neglect of the diffuser and tip clearance in numerical simulations. Moreover, the leakage, roughness, and mechanical losses are inevitable in the experimental test. On the other hand, the steady-state analysis results show the highest absolute value among the data. The unsteady-state analysis results are distributed closer to the
experimental data than the steady-state analysis results. Hence, the unsteady-state analysis reflects the simulation of the internal flow and phenomena during the pump operation according to the time variation, which means that the more accurate prediction is performed. Each non-cavity state, that is, the absolute value at the 0% head drop where the inlet pressure is greatly high, also shows the difference in the same order. Figure 5 shows the blade loading distribution at the shroud span for each model. The pressure coefficient ($C_p$) is defined as follow:

$$C_p = \frac{P - P_1}{\frac{1}{2}\rho W_1^2}$$

where $P$, $P_1$, and $W_1$ denote the static pressure, inlet total pressure, and relative velocity at the impeller inlet, respectively. First, comparing the results of steady- and unsteady-state analysis, it is possible to confirm the difference of absolute value at the 0% head drop. The blade loading distribution is significantly different on the pressure surface near the impeller TE. The predicted pressure value in steady-state analysis is higher than the unsteady-state results both type A and B. The head level differences in the steady- and unsteady-state analysis in Figure 4 are due to the difference in predictions.
on the pressure near impeller TE. The distribution of pressure fluctuations can be also observed. Type A shows the same distribution over time, while type B cannot identify a constant pattern from LE to TE. The constant pattern from LE to TE. The abbreviation T in Figure 5 is the time for one revolution of the impeller (0.125 s). Meanwhile, this study is analyzed at the design point, which is located near the best efficiency point (BEP) as shown in Figure 6. The design point is indicated with a dotted line.

As the detailed analysis for each model, type A and B tend to differ in their head drop gradient. The head of type A maintains a non-cavity value despite the decrease in inlet pressure, and then shows a sharp slope near the 3% head drop. The head of type B, however, shows a tendency for the gradual head drop from about \( \sigma = 2 \). From the results, type B reached the 3% head drop point faster and had worse cavitation performance. In particular, Figures 4(a) and (b) show the pressure fluctuation ranges in the unsteady-state results. The averaged value of the last (7th) revolution was represented as a green square, and the maximum and minimum values of the last (7th) revolution were represented as the range bars. There are little pressure fluctuations for all points of type A while the wide range of pressure fluctuations are appeared in type B starting from \( \sigma = 0.78 \).

The magnitude of the pressure fluctuation has a direct effect on the bubbles oscillation (wiggle) as shown in Figure 7. At \( \sigma = 0.78 \), a significant difference was observed with the vapor volume fraction averaged circumferential area at the impeller LE of type A and B. First, the bubbles of type A does not change over time. Moreover, it is fully consistent with the data of steady-state analysis. The amount of bubble is concentrated near the shroud. However, the bubbles of type B can be seen that the amount is different with time variation. This means that the bubbles pattern and shape are changing while the impeller is rotating. Additionally, the bubbles are perceived as another blockage in the point of view for the working fluid, water, so those phenomena can affect the hydraulic performance of a pump. However, it cannot be concluded sufficiently that the cavitation characteristics associated with bubbles oscillation influenced the head drop.

5. Conclusion
The cavitation characteristics for the head drop in two mixed-flow pumps were investigated with the steady- and unsteady-state numerical analysis. In the simulation for the cavitation, the unsteady-state analysis can obtain a more accurate prediction than the steady-state analysis. The pumps with good cavitation characteristics have a sharp decrease in the head near the 3% head drop point. The head of the pumps which performed with poor cavitation characteristics decreases gradually as the inlet pressure decreases, and reaches faster at the 3% head drop point. In addition, the pumps with poor cavitation characteristics exhibit wide pressure fluctuations from the 3% head drop, which is directly related to the bubbles pattern.
References

[1] Higashi S, Yoshida Y and Tsujimoto Y 2002 Tip leakage vortex cavitation from the tip clearance of a single hydrofoil JSME Int. J. B-Fluid Therm. Eng 45 pp 662–671

[2] Lu J, Yuan S, Parameswaran S, Yuan J, Ren X and Si Q 2017 Investigation on the vibration and flow instabilities induced by cavitation in a centrifugal pump Advances in Mechanical Engineering 9(4)

[3] Grist E 1998 Cavitation and the Centrifugal Pump (USA: Taylor & Francis)

[4] Fu Y, Yuan J, Yuan S, Pace G, d’Agostino L, Huang P and Li X 2014 Numerical and experimental analysis of flow phenomena in a centrifugal pump operating under low flow rates ASME Journal of Fluids Engineering 137(1) 011102–12

[5] Li H, Shen Z, Pedersen N E and Jacobsen C B 2019 Experimental and unsteady numerical research of a high-specific-speed pump for part-load cavitation instability Advances in Mechanical Engineering 11(3)

[6] Kim Y I, Kim S, Yang H M, Lee K Y and Choi Y S 2019 Analysis of internal flow and cavitation characteristics for a mixed-flow pump with various blade thickness effects Journal of Mechanical Science and Technology 33(7) pp 3333-3344

[7] Gulich J F 2008 Centrifugal Pumps, (Germany: Springer-Verlag Berlin Heidelberg)

[8] Tang F and Li J W 2010 Numerical simulation of rotating cavitation in a liquid hydrogen pump inducer Proceedings of the 13th Asian Congress of Fluid Mechanics Bangladesh

[9] Ding H, Visser FC, Jiang Y and Furlanczyk M 2011 Demonstration and validation of a 3D CFD simulation tool predicting pump performance and cavitation for industrial applications ASME Journal of Fluids Engineering 133(1) 011101–14.

[10] Liu X, Hu Q, Wang H, Jiang Q and Shi G 2018 Characteristics of unsteady excitation induced by cavitation in axial-flow oil–gas multiphase pumps Advances in Mechanical Engineering 10(4)

[11] Lee S N, Tak N I and Noh J M 2013 Heat transfer prediction in pipe flow by the wall function of SST turbulence model KSCFE Conference pp 355–358

[12] Lee Y G, Yuk J H and Kang M H 2004 Flow analysis of fluid machinery using CFX pressure-based coupled and various turbulence model The KSFM Journal of Fluid Machinery 7(5) pp 82–90