Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 26-27, 2023

CFD Analysis of the Cavitation Flow Features for the Centrifugal Pump with Different Numbers of Blade and Blade Angle

Gong Hee Lee[a*], Yong Kab Lee[b]
[a]*System Evaluation Department, Korea Institute of Nuclear Safety, Daejeon, 34142, Korea*
[b]*SIMEX, Seoul, 08511, Korea*

[*]*Corresponding author: ghlee@kins.re.kr*

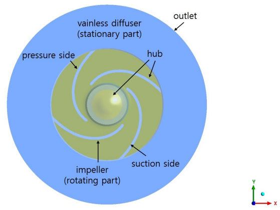
Keywords : blade angle, cavitation, centrifugal pump, computational fluid dynamics, in-service testing

1. Introduction

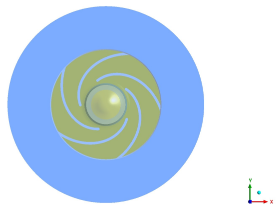
Domestic nuclear power plant (NPP) operators have conducted in-service testing (IST) to confirm the safety functions of safety-related pumps and to monitor the degree of venerability over time during reactor operation [1]. One of the representative IST-related pump types, a centrifugal pump, is commonly used to perform the safety functions. When the suction pressure of a centrifugal pump decreases, the cavitation flow may occur to give rise to noise, vibration, performance degradation, and the impeller blade can be damaged due to erosion caused by bubble collapse [1]. Various numerical studies were conducted to examine the cavitation flow behavior inside the centrifugal pump [2,3]. In this study, the computational fluid dynamics (CFD) analysis of the cavitation flow inside a single-stage centrifugal pump at the Pfleiderer Institute [4] was performed and the change in the cavitation flow pattern depending on the numbers of blade and the blade angle was identified. The computational results for the steady state cavitation flow inside the TFA centrifugal pump with a scale-down impeller blade (scale factor = 0.5) can be found in the author's previous studies [5].

2. Analysis Model

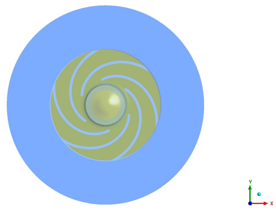
Fig. 1 shows a schematic diagram of the analysis model. A vaneless radial diffuser was used to produce the uniform pressure distribution at the impeller outlet [1,4]. The counter-rotating impeller blade consisted of two simple circular arcs to retain an approximately two-dimensional flow field and offer superior accessibility for measurement [1,4]. Three types of blade with different numbers of blade and blade angle were considered. Geometric specifications of the analysis model are summarized in Table I. Since the blade showed the geometrical symmetry, the rotational periodic condition was applied to one blade to reduce the calculation time. Real shroud cavity was considered to improve the prediction accuracy for the pump flow field. The working fluid was assumed to be 20 °C of water [1].



(a) Type A



(b) Type B



(c) Type C

Fig. 1. Analysis model (top view).

Table I: Geometrical specification of an analysis model

Items	Unit	Type A	Type B	Type C
Blade shape	-	2 circular arc	2 circular arc	2 circular arc
Inlet diameter	mm	260	260	260
Outlet diameter	mm	556	556	556
Inlet blade angle	Deg.	**17**	**19**	**20**
Outlet blade angle	Deg.	**30**	**23**	**19**
Passage width	mm	46	46	46
Number of blades	-	**4**	**5**	**6**
Specific speed	-	27.5	27.5	27.5
Blade thickness	mm	13	13	13
Rotating speed	Hz	9	9	9

3. Numerical Modeling

In this study, the cavitation flow inside the centrifugal pump was calculated under steady, incompressible, turbulent, and multi-phase flow conditions using ANSYS CFX 2021R1. For reference, the numerical methods and boundary conditions used in this study was summarized in Table II.

Table II: Numerical methods and boundary conditions for flow analysis

Numerical methods		Note
Discretization accuracy for convection term	Momentum eqn.	High resolution
	Turbulence eqn.	High resolution
Interphase transfer model		Mixture
Cavitation model		Rayleigh-Plesset
Turbulence model		SST k-ω
Near wall treatment		Automatic wall treatment
Impeller-diffuser boundary interaction		Stage (Mixing plane)
Convergence criteria		$< 10^{-4}$
Boundary conditions		**Note**
Inlet	Flow rate	420 m^3/h
	Turbulence	medium intensity (5%)
	Liquid volume fraction	1.0
Outlet		opening option & static pressure
Wall		no-slip & smooth wall

As shown in Fig. 2, the unstructured hybrid (consisting of hexahedral, tetrahedral and wedges type) grid system was used. Total elements number was about 2×10^6 and the denser grid was distributed near the hub, blade, and shroud wall [1].

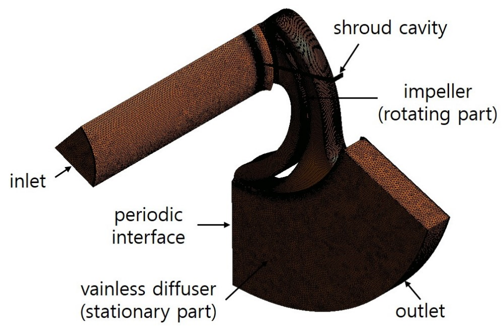
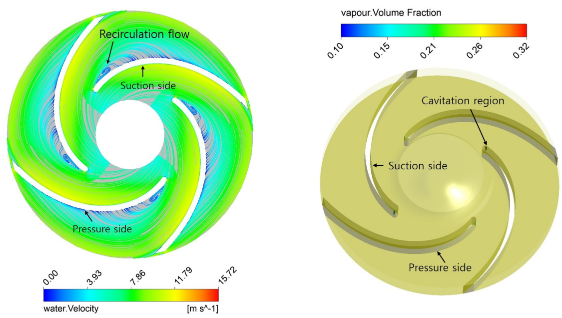


Fig. 2. Grid system (Type C, one passage).

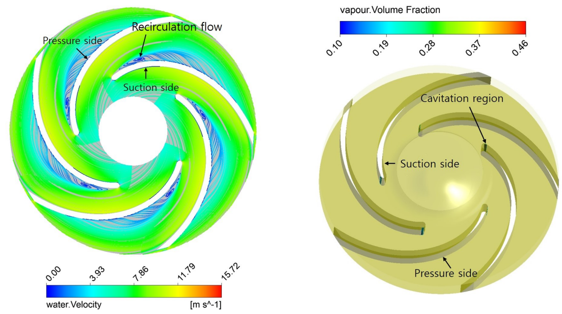
4. Results and Discussion

Fig. 3 shows the streamline at the midspan and vapor volume fraction distribution at 3% head drop and the design flow rate of 420 m^3/h. For other cases except for Type C, recirculation flow occurred in the pressure surface near the blade leading edge. The above-mentioned recirculation flow may act as a blockage in the flow passage between the blades. On the other hand, weak cavitation flow generated in the pressure surface near the blade leading edge. As the number of blades and the inlet blade angle increased, the cavitation flow region slightly expanded.

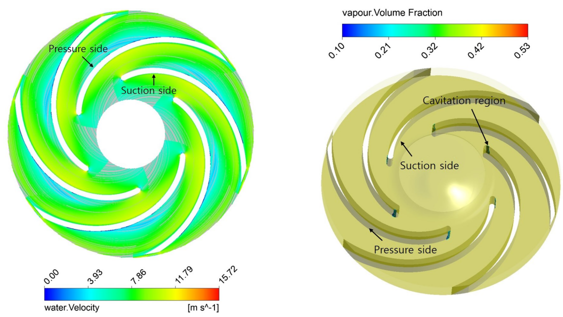
Fig. 4 shows the streamline at the midspan and vapor volume fraction distribution at 10% head drop and the design flow rate of 420 m^3/h.



(a) Type A



(b) Type B



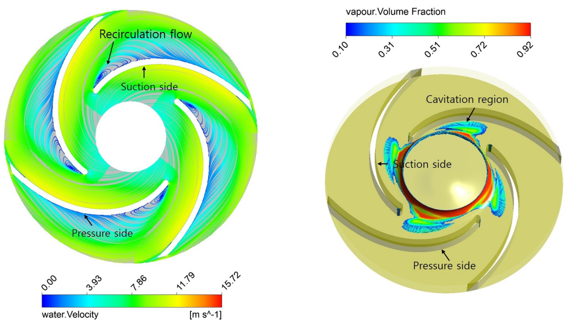
(c) Type C

Fig. 3. Streamline at the midspan (left) and vapor volume fraction distribution (right) : 3% head drop.

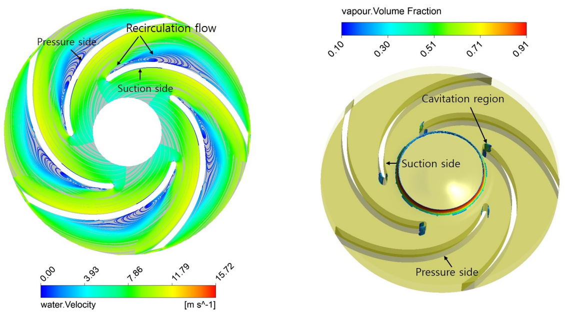
Unlike that of 3% head drop, for all cases, recirculation flow was found in the blade pressure surface. For Type B and Type C, weak cavitation flow also occurred in the suction surface near the blade leading edge. The cavitation flow region near the shroud significantly expanded as the number of blades and the inlet blade angle decreased.

5. Conclusions

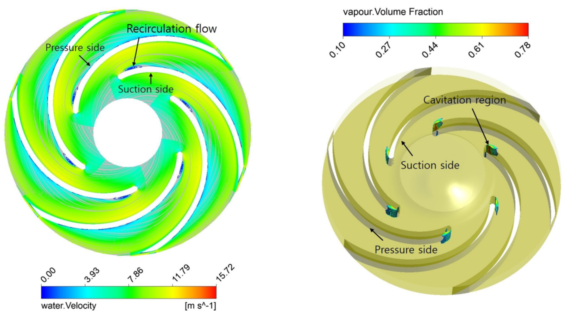
From the CFD analysis of the cavitation flow inside a single-stage centrifugal pump at the Pfleiderer Institute, it was found that the cavitation flow pattern changed depending on the numbers of blade and the blade angle. Additional simulation for the different flow rate and head drop and the supplementary results (pressure distribution, Net Positive Suction Head, etc.) will be shown in a separate paper.



(a) Type A



(b) Type B



(c) Type C

Fig. 4. Streamline at the midspan (left) and vapor volume fraction distribution (right) : 10% head drop.

DISCLAIMER

The opinions expressed in this paper are those of the author and not necessarily those of the Korea Institute of Nuclear Safety (KINS). Any information presented here should not be interpreted as official KINS policy or guidance.

ACKNOWLEGEMENT

This work was supported by the Nuclear Safety Research Program through the Korea Foundation Of Nuclear Safety (KOFONS) using the financial resource granted by the Nuclear Safety and Security Commission (NSSC) of the Republic of Korea (No. 1805007). This work was also supported by the Korea Institute of Nuclear Safety (A3FD23030 & A6FD23020).

REFERENCES

[1] G. H. Lee, Y. G. Lee, Numerical Study for the Cavitation Flow inside the PFI Centrifugal Pump, ACFD2022, Oct.16-19, 2022, Jeju, Republic of Korea.

[2] R. B. Medvitz, R. F. Kunz, D. A. Boger, J. W. Lindau, A. M. Yocum, L. L. Pauley, Performance Analysis of Cavitating Flow in Centrifugal Pumps using Multiphase CFD, Journal of Fluid Engineering, Vol.124, p. 377, 2002.

[3] B. Pouffary, R. F. Patella, J.-L. Reboud, P.-A. Lambert, Numerical Simulation of 3D Cavitating Flows: Analysis of Cavitation Head Drop in Turbomachinery, Journal of Fluid Engineering, Vol.130, 061301, p. 1, 2008.

[4] M. Hofmann, B. Stoffel, J. Friedichs, G. Kosyna, Similarities and Geometrical Effects on Rotating Cavitation in two scaled Centrifugal Pumps, 4th International Symposium on Cavitation (CAV 2001), June 20-23, 2001, California, USA.

[5] G. H. Lee, Y. G. Lee, Numerical Analysis for the Cavitation Flow inside the TFA Centrifugal Pump using Various RANS-based Turbulence Models, Korean Energy Society Autumn Meeting, Oct.19-21, 2022, Yangyang, Republic of Korea.