Preliminary CFD Analysis for the Rotating Cavitation Flow inside a Centrifugal Pump

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 : centrifugal pump, computational fluid dynamics, in-service testing, rotating cavitation, rotor-stator

1. Introduction

A centrifugal pump is commonly used to perform the safety functions in the nuclear power plant. When cavitation flow occurs in a centrifugal pump, it may cause the noise, vibration, performance degradation, and impeller blade damage.

A rotating cavitation, i.e., the alternate cavitation propagating from blade to blade similar to the rotating stall in a compressor can be found in the flow range where the pump performance curve (head vs. flowrate) has a negative slope [1]. In this flow regime, blade forces may be quite high and the various mechanical parts of a pump may be easily excited to resonance [2].

In this study, preliminary computational fluid dynamics (CFD) analysis for the rotating cavitation flow inside a single-stage centrifugal pump at the Pfleiderer Institute (PFI) [3] was performed by applying the two types of frame change/mixing models available in ANSYS CFX and then simulation results were compared.

2. Analysis Model

Fig. 1 shows a schematic diagram of the analysis model. A vaneless radial diffuser was used to obtain the uniform pressure distribution at the impeller outlet [3,4]. The counter-rotating impeller blade consisted of two simple circular arcs to retain an approximately two-dimensional flow field [3,4]. Geometric specifications of the analysis model are summarized in Table I.

Because a rotating cavitation generally shows the asymmetrical pattern, the full flow passage is considered instead of applying the rotational periodic condition to one blade.

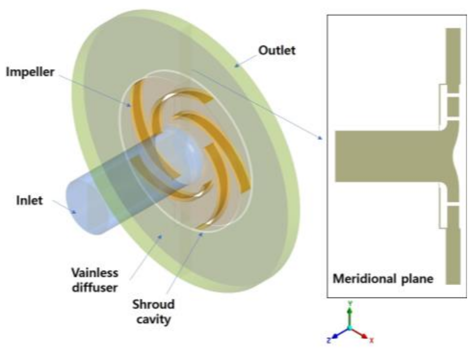


Fig. 1. Analysis model.

Table I: Geometrical specification of an analysis model

Parameters	Contents	Parameters	Contents
Blade shape	2-circular arc	Passage width	46 mm
Scale factor	1.0	Number of blades	5
Inlet diameter	260 mm	Specific speed	27.5
Outlet diameter	556 mm	Blade thickness	13 mm
Blade inlet angle	19°	Impeller rotating speed	9 Hz
Blade outlet angle	23°	Rotation direction	Counter-clockwise

A shroud cavity was also considered to improve the prediction accuracy for the pump flow field [5]. The working fluid was assumed to be 20 °C of water [5].

3. Numerical Modeling

In this study, a rotating cavitation flow inside the PFI centrifugal pump was calculated under transient, incompressible, turbulent, and multi-phase flow conditions using ANSYS CFX 2022R1. For reference, the numerical methods and boundary conditions used in this study were briefly explained 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
Frame change/mixing model		Frozen rotor or Transient rotor stator
Convergence criteria		$< 10^{-4}$
Boundary conditions		**Note**
Inlet	Flow rate	231 m^3/h ($0.55Q_d$)
	Turbulence	medium intensity (5%)
	Liquid volume fraction	1.0
Outlet		static pressure
Wall		no-slip & smooth wall

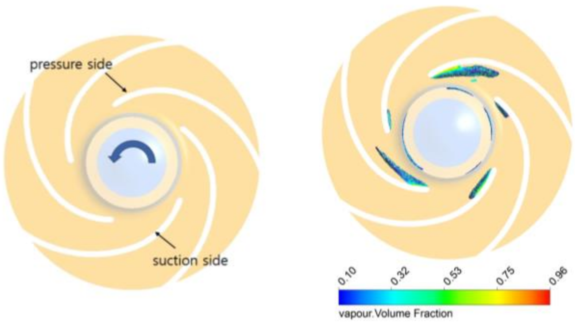
The unstructured hybrid (consisting of tetrahedral, wedges and pyramids type) grid system was used. Total elements number was about 1.17×10^7 and the denser grid was distributed near the hub, blade, and shroud wall [5].

Two types of frame change/mixing models available in ANSYS CFX, i.e., Frozen Rotor and Transient Rotor-Stator, were applied in this study. Frozen Rotor model requires relatively less computational resources than other models, but it may have the modeling errors when the quasi-steady assumption does not apply. Transient Rotor-Stator model can predict the real transient flow interaction between a rotor and stator passage.

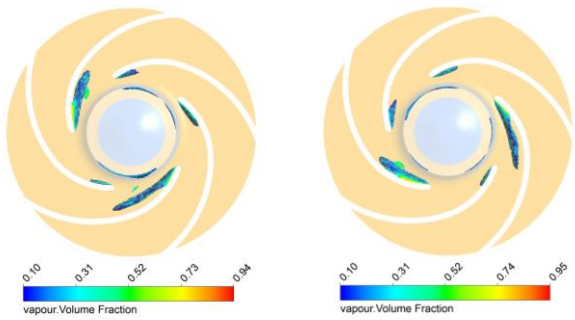
4. Results and Discussion

Fig. 2 shows the predicted vapor volume fraction distribution by applying the Transient Rotor-Stator model. The simulation was conducted at the flow rate of 231 m^3/h (corresponding to 55% of design flow rate Q_d) and cavitation number of 0.6. For reference, the location of each blade is same regardless of the revolution numbers. The cavitation flow was generated from the suction surface near the blade leading edge and showed asymmetrical pattern, e.g., different cavity size. The cavities of either small or large sizes were alternatively observed.

Fig. 3 shows the predicted vapor volume fraction distribution by applying the Frozen Rotor model. The simulation condition was the same as that of Fig. 2, but an additional calculation for more revolutions was conducted to find the existence of the rotating cavitation flow. Unlike the results shown in Fig. 2, the cavity sizes in all flow passages were nearly same. Therefore, the Frozen Rotor model is not recommended for the accurate prediction of the rotating cavitation flow.

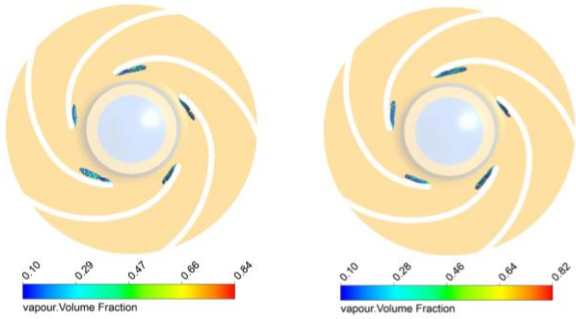


(a) Plane view (Impeller) (b) 8th revolution



(c) 9th revolution (d) 10th revolution

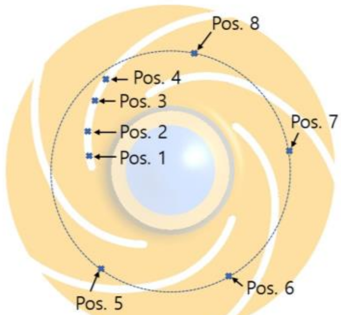
Fig. 2. Vapor volume fraction distribution (Transient Rotor-Stator model).



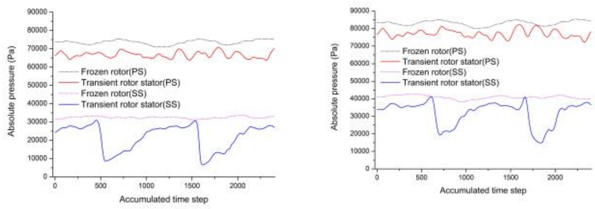
(a) 13th revolution (b) 14th revolution

Fig. 3. Vapor volume fraction distribution (Frozen Rotor model).

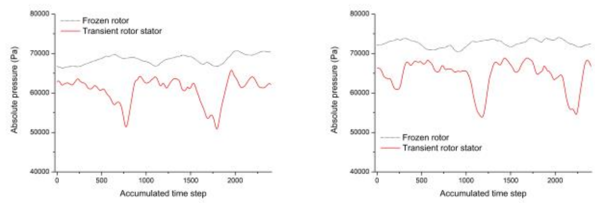
Fig. 4 shows the unsteady pressure at the selected monitoring locations. Pos. 1~4 denote the blade pressure (suction/pressure side) taps and Pos. 5~8 indicate the hub pressure taps. The Transient Rotor-Stator model showed more significant pressure variations on the blade suction side and hub taps than the Frozen Rotor model. One of the potential contributors may be the existence of the rotating cavitation flow.



(a) Pressure monitoring locations



(b) Pos. 3 (c) Pos. 4



(d) Pos. 7 (e) Pos. 8

Fig. 4. Unsteady pressure at the selected monitoring locations.

5. Conclusions

From the preliminary CFD analysis for the rotating cavitation flow inside the PFI single-stage centrifugal pump, the following main conclusions can be obtained:

(1) It was found that a rotating cavitation flow could be numerically captured by applying Transient Rotor Stator model. The cavitation flow was generated from the suction surface near the blade leading edge and showed asymmetrical pattern. The cavities of either small or large sizes were alternatively observed.

(2) Due to the existence of the rotating cavitation flow, the Transient Rotor-Stator model showed more significant pressure variations on the blade suction side and hub taps than the Frozen Rotor model.

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 (A3FD24030 & A6FD24021).

REFERENCES

[1] Y. Tsujimoto, On Rotating Cavitation, Proc. KFMA, p. 13, 1997.
[2] A. J. Acosta, An Experimental Study of Cavitating Inducer, Proc. 2nd Symp. Naval Hydrodynamics, ONR/ACR-38, p.537, 1958.
[3] 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.
[4] 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.
[5] G. H. Lee, Y. G. Lee, CFD Analysis of the Cavitation Flow Features for the Centrifugal Pump with Different Numbers of Blade and Blade Angle, Transactions of the Korean Nuclear Society Autumn Meeting, Oct. 26-27, 2023, Gyeongju, Korea.