Cavitation flows in a Kaplan turbine

Ryo Amano[1]*, James Smith[2]

ISROMAC 2017

International Symposium on Transport Phenomena and Dynamics of Rotating Machinery

Hawaii, Maui

December 16-21 2017

Abstract

One of the challenging problems in a hydro machine is cavitation which is an aggravating problem in fluid flow applications. Detrimental effects are ranging from material pitting/cracking and full system damage according to the amplitude of implosions. Such occurrence could cause costing much in maintenance or complete replacement of the damaged body. This study demonstrates the conditions when the cavitation phenomena in Kaplan turbine can occur. A large-eddy simulation (LES) is made to predict a cavitating flow through a small axial hydro-turbine. Interactions between three fluids (liquid water and water vapor) were analyzed using the Volume of Fluid (VOF) multiphase, computations. The study clarifies the conditions of onset of cavitation.

Keywords

Hydro turbines, cavitation

[1]*Department of Mechanical Engineering, University of Wisconsin-Milwaukee, WI, United States*

***Corresponding author**: amano@uwm.edu

INTRODUCTION

It was reported by Alhashan and Addali [1] that, in the saltwater, there were no large individual bubbles. However, small bubbles were generated and persisted in the container for a very long time. Although the mechanism of bubble formation is the same in salt water and fresh water, there is a difference in the size of the cavitation bubbles. As Gowing [2] suggested, smaller bubbles play a significant role in incipient cavitation. Blanchard et al. [3] noted that the bubbles coalesce in tap water more than in sea water. Furthermore, the bubbles produced in sea water are smaller and continue longer than in pure water [4,5], and demonstrated that there is a significant difference between bubble cavitation in salt water and fresh water. It was further observed that the acoustic emission of the bubble cavitation in fresh water was lower than that produced by salt water as shown in Fig. 1 [6]. Furthermore, they noted that small bubbles produce higher acoustic emission compared to large bubbles. Bubbles of similar size produced similar acoustic emission regardless whether they are in salt water or fresh water, and the chemical difference of the water does not appear to influence the acoustics directly.

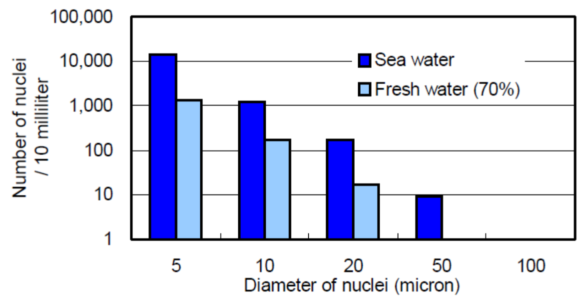

Figure 1. comparison of cavitation bubble size between seawater and freshwater [8].

The UWM team has been investigating on how to reduce the cavitation formation for freshwater hydro turbine, and recently identified that an injection of small amount of air into cavitation cloud can reduce the formation by 70%. With the knowledge developed so far, we further investigated the effect of aeration technique applying to a hydro turbine with salt-water usage and designed a manifold that reduces cavitation in both fresh and seawater environments.

.

1. METHODS

1.1 Experiment

To identify a technique that can significantly reduce cavitation formation in a hydro turbine with both freshwater and seawater, the project aims to analyze the cavitation bubble formation process in a hydro-turbine unit and the interaction mechanism with the injected air with the bubbly flow of cavitation. The cavitation bubbles on a blade surface, as shown in Fig. 2, needs to be investigated with the air interactions. Fig.2(a) shows caviation on plate surface (Cosford [7]). It is well recognized that severe damage might occur on a hydro machine surface when these bubbles collapse near each other, referred to as “cloud cavitation” that our picture taken at UWM Hydro Research Lab (Fig.2(b)) and Arndt’s data [8] (Fig.2(c)). This “cloud cavitation,” phenomenon creates very adverse effects as a result of cloud formation due to collation. Such a phenomenon might cause more severe damage on the blade surfaces of any hydro turbomachines and larger vibration since a cloud cavitation accompanies very high-pressure pulses within the cloud and radiates away during the collapse process. Within the clouds, such pulses can move up as high as 30 atmospheric pressures within 0.1msec [9]. Cavitation structures associated with the pressure pulses is a key factor to investigate for reducing the cavitation damage suffering from cloud shock.

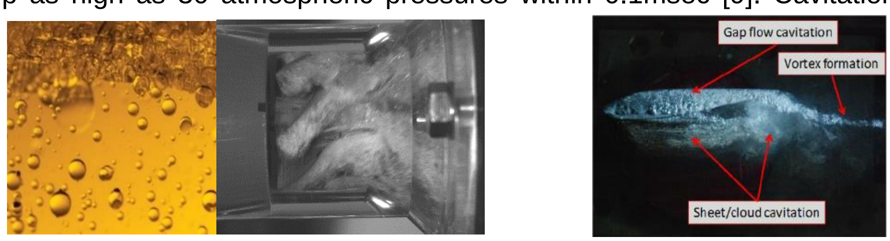

Figure. 2. High-speed camera views. (a) Left: from Cosford; (b) Middle: taken at UWM Hydraulic Lab (4000 fps with 0.5 m focus); (c) Right: Cloud cavitation (Arndt).

The research team performed computational fluid dynamics (CFD) and experiments with both freshwater and saltwater, and studied the onset of cavitation where on the runner area those phenomena might occur. The group also investigated the areas where the pressure drops under a certain level to cause vaporization. Then, using both the CFD and experimentation, cavitation was identified with the variation of the vapor volume fraction (VVF) in the surface area of the runner blade surface where the highest velocity and, thus, very low pressure were created in such areas.

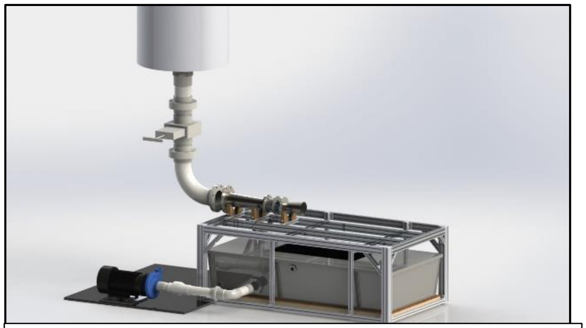

Figure 3. Test facility being built at Global Water Center at UWM.

1.2 CFD Research

The research team has much of the experience on CFD work on both stator and rotor for Kaplan hydro turbine runner simulations [10,11]. Regarding the simulation models, the Reynolds Averaged Navier-Stokes (RANS) models provide results for mean quantities with engineering accuracy at moderate cost for a wide range of flows. However, for non-equilibrium flows, those involving separation, recirculation, rotation and two-phase phenomenon, and flows dominated by large-scale vortical structures, the computed average quantities are less than satisfactorily predicted with the RANS models.

Large Eddy Simulation (LES) generally performs well and bears few modeling uncertainties. Furthermore, LES, by construction, provides unsteady data for a large portion of large-scale spectra, which are indispensable in many cases: determination of unsteady forces, fluid-structure coupling, identification of aerodynamic sources of sound, and phase-resolved multiphase flow, to name but a few. Among the current turbulence simulation techniques, LES is well recognized as a promising way to predict complex turbulent flows. The geometries of several arrangements were tested. Numerical simulation was performed using a 3-D LES code combined with the two-phase flow model through the stator/rotor path in a Kaplan turbine.

The governing equations were discretized on a combination of the structured/unstructured grid using a second-order upwind difference scheme. Investigating erosion and the noise effects using the air injection system are the main tasks for this study.

In the CFD model, a pulsed air injection unit was implemented and a parametric study with various air mass flow rates with different turbine operating conditions for seawater were conducted.

Furthermore, the full cavitation model was used in the simulations that include many effects such as unsteadiness, vibration, different salt percentages, and turbulence. The research team investigated several levels of cavitation, among which important types include: (1) “Incipient cavitation”, (2) “Constant cavitation” and (3) “Choked cavitation” for several hydro machines. The test setup, as shown in Fig. 3, were constructed in the Global Water Center, a 98,000 sq. ft. facility housing water-centric research facilities for universities, existing water-related companies, and accelerator space for emerging water technology companies.

2. RESULTS AND DISCUSSION

Baseline case (i.e. cavitation only) was firstly investigated to check the initiation location and propagation of the vapor cloud. The steady state is reached at 0.06s; latter times were considered for averaging the steady properties. The minimum pressure coefficient and cavitation number, calculated from the velocity and pressure entering the rotor, are found to be (-Cp_{min}= 6.04, and σ= 6.08). Such status leads to the formation of vapor at two main locations: the rotor blade leading edge extending along the chord, and the hub-blades intersection where the vapor spreads over part of the hub and the span of the blades. Cavitation behavior is randomly fluctuating because of the vapor cloud cyclic nature (formation, separation, and collapse), and the relative position of the blade to the incoming flow from the stator. Figure 2 can show the local and temporal variation in the distribution of the vapor volume fraction (VVF) at the rotor.

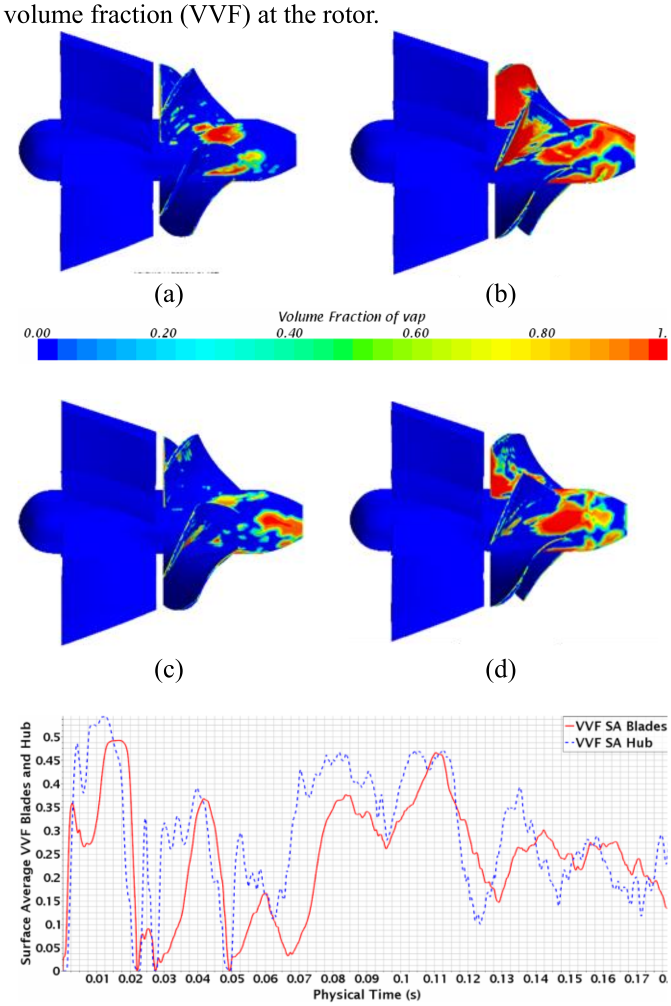

Figure 1: VVF distribution (non-Blue colors) at different times (a) 0.064s, (b) 0.1s, (c) 0.128s, (d) 0.18s, and (e) time history for the surface averaged VVF over the blades and the hub

When air enters the system like a jet, it is affected by the three motion components: radially inward (i.e. penetrating the other fluids towards the turbine axis) due to the pressurization, axial flow with the liquid water, and a rotational with the blades cycle. Accordingly, the location of air is variant with space and time. The air content is added as a factor in changing the shape of the vapor cluster on the rotor. Figure 4 displays the air and vapor volume fraction on a mid- sectional plane after 0.18s. The domination of air at one location (e.g. dashed circles) diminishes the existence of vapor at the same place, which indicates the merit of air injection. This leads to further analysis about the reduction in vapor content.

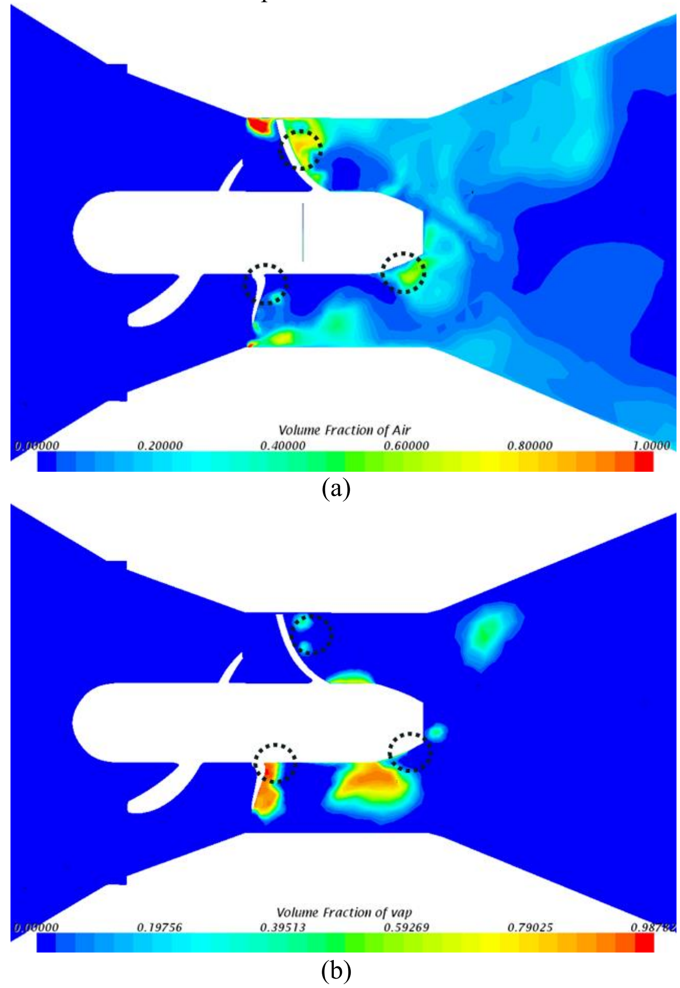

Figure 2: Volume fraction distribution for (a) air, and (b) water vapor after 0.18s. dashed circles show some locations of high and low vapor presence in relation with the air existence at the same place

Time averaging the VVF on the rotor parts gives a unique number for each case. The surface and time averaged VVF number is directly related to flow behavior and the interaction of air distribution over the rotor. VVF is used to compare the amount of vapor created on the blades and hub in each case (no-air, 1 port, 12 ports, and 6 ports). Table 1 shows the VVF and expected mechanical power (P) for the cases of no-air, 1 air-port, and 12 air-ports. The percentage change (reduction in VVF and increase in P) is based on the no-air result as a reference. Air-water flow rate ratio is calculated in each case to point to the effect of increase the amount of injected air.

CONCLUSIONS

Studying the effect of air on a cavitating flow in turbomachinery, a time-dependent CFD case was made for a 3-inch Kaplan hydro-turbine undergoing cavitation, which is developed to cases of pressurized air injection as a jet-in-cross-flow. Once initial trials showed significance, different patterns of injection were suggested to check the effect of the increased air mass flow rate and test the effectiveness of the linear arrangement. By surface averaging over the rotor surfaces, followed by time averaging in the steady state period, vapor volume fraction was represented as one value for each case. VVF values with the mechanical output power (P) were compared for the proposed cases, presenting a VVF reduction of large increase and modest power increase with raising air-water flow rate ratio. Moreover, side injection system shows effectiveness in minimizing the vapor with substantial increase with a modest power rise. Future analysis are planned for the optimal position of the ports and orientation of injection.

REFERENCES

1 Alhashan, T. and Addali, A., “The Effect of Salt Water on Bubble Formation during Pool Boiling Using Acoustic Emission Technique,” IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 13, Issue 5 Ver. VIII (Sep. - Oct. 2016), PP 51-56 www.iosrjournals.org

2 Gowing S. Dissolving of Bubble in a Liquid. DTNSRDC Report. 1987; 6: 87.

3 Blanchard, D. C., and Woodcock AH. Bubble Formation in Modification in the Sea and its Meteorological Signifcance. Tellus. 1957; vol.9, NO.: pp. 145–158.

4 S. Ceccio, S. Gowing YTS. The Effects of Salt Water on Bubble Cavitation. Journal of Fluids Engineering. Vol. 119: 155–163.

5 Abe T. On the Stable Formation of Sea Water in Seas. journal of the oceanographical society of Japan. 1962; 20: pp. 242–250.

6 Kamiirisa,H., “The Effect of Water Quality Characteristics on Cavitation Noise,” CAV 2001, Session A2.004, 2001

7 J. Cosford, “How did my hydraulic system get so dirty,” Fluid Power World, October 8, 2016. http://www.fluidpowerworld.com/author/jcosford/ 9visited on 10/9/2016)

8 Arndt, R.E.A., “Cavitation research from an international perspective,” IOP Conference Series Earth and Environmental Science November 2012, DOI: 10.1088/1755-1315/15/1/012002

9 Brennen, C., Colonius, T., Wang, Y-C., and Preston, A., “Cloud Cavitation Phenomena,” Twenty-Second Symposium on Naval Hydrodynamics, 2000.

10 Amano, R.S., ElGammal, T., Yen, Y., Millevolte, J., Mueller, R., and Lequesne, B., “Numerical Investigation on Intake Tube Dimensions of Low Head Kaplan Turbine,” FEDSM2016-7569, ASME Fluids Engineering Division Summer Meeting, July 10-14, 2016, Washington D.C.

11 Amano, R.S., Yen, ElGammal, T., Y., Millevolte, Lequesne, B.J., and Mueller, R., “Numerical Investigation on Intake Tube Dimensions of Low Head Kaplan Turbine,” FEDSM2016-7575, ASME Fluids Engineering Division Summer Meeting, July 10-14, 2016, Washington D.C.