Numerical Analysis of Water Vortex Formation for the Water Vortex Power Plant

Tze Cheng Kueh, Shiao Lin Beh, Dirk Rilling, and Yongson Ooi

***Abstract*—The behavior of the water vortex is interested in designation of turbine blade for Water Vortex Power Plant (WVPP). This study uses Xflow, a commercialize CFD code base on Lagrangian approach, which provides the details of water vortex formation without much difficulty as discretization of flow domain is not required at all. The boundary conditions of the CFD models are applied based on the experiment setup. Two conditions were investigated, with two different holes size for water discharge. The result in first condition shows the vortex height agree each other in experiment and CFD. For second condition, the final vortex height of the CFD model is different from the experiment result. As the discharge hole is getting larger, more turbulent flow has set in and causing more errors in the CFD model in predicting the water vortex formation. However, the CFD model can be improved by better turbulent modeling.**

***Index Terms*—CFD, water vortex, turbine.**

I. Introduction

A. Background

Water vortex is a phenomenon where water flow in a swirl motion, always described by cylindrical coordinate, with tangential, radial, and axial axis. In 1858, Rankine already published his study on water vortex by introduced mathematical model for the tangential velocity of the water vortex [1]. Afterward, other researchers, Odgaard [2], Hite and Mih [3], C. Yun-Liang *et al*. [4] also studied water vortex and tried to modify and to improve the mathematical model of the water vortex. Their focus is on the vortex generated with the hydraulic intake in a free surface flow, where the formation of vortex is undesirable as it will decrease the efficiency and damage the devices.

In process industry, water vortex also been studied for the purpose of mixing process. The mixing process uses a rotor to rotate the liquid inside either a baffled or unbaffled vessel. The knowledge of vortex shape is important in this area for design purpose. Torre *et al*. [5] use CFD to model the free surface in a partially baffled vessel while Busciglio *et al*. [6] investigate the vortex shape in unbaffled stirred vessel by digital image analysis.

There is a machine uses water vortex to generate power. It is water vortex power plant (WVPP). WVPP is a turbine that functions with a similar way to Kaplan turbine, which is an axial type of turbine uses the swirl flow of water to generate power. However, Kaplan turbine uses guide vane to generate the swirl motion in a closed vessel while the swirl motion of water in WVPP comes from the naturally organized water vortex on a free surface. The idea of WVPP comes from a forester, Viktor Schauberger [7] who see water vortex motion as an implosion phenomenon that creates, develops, purifies, and grows. He promoted to use more centripetal motion in technology to replace centrifugal motion like explosion and combustion which is said to against natural.

The conditions of water vortex in WVPP are different from the two cases mentioned above. For water vortex of hydraulic intake, the water vortex is generated in at free flow condition with an intake, either horizontal or vertical at the bottom of the water. This type of water vortex is not limited by the ground and wall boundary. The proposed mathematical model describes the velocities of the vortex to a function inversely to the radius and the axial velocity to be a linear function to the depth of water. Meaning that, water vortex in this model having its velocity extended to an unlimited radius, and the axial velocity is keep increasing from the water surface to the bottom. This condition is not suit to be applied to a vortex in a container which is bounded by its wall and ground.

In the case of mixing process, the water vortex is generated in a vessel, similar to the WVPP condition. However, the vortex generation is caused by the external force, usually an impeller driven by a motor. The vortex height is always reach the depth of the impeller itself. However, for WVPP, the water vortex is naturally generated by the water inside a basin with a bottom hole discharging the water. The vortex occurred sometimes will form a thoroughly air core from water surface to the bottom hole. In [7], Schauberger gave a picture of this self-organized water vortex motion, named it as toroidal vortex flow.

B. Problem Statement and Objective

For the design purpose of WVPP, more details on self-organized water vortex inside a basin e.g., vortex profile, core radius, and velocities are needed. To obtain these information, it is time and cost consuming to measure it from the experiment. Hence, as other hydrodynamic problems, computational fluid dynamics (CFD) is a good idea to model the system and obtain the information of the system. The objectives of the paper are to study the self-organized vortex in a round vessel by CFD and verify it by comparing with an experimental setup basin.

Manuscript received December 20, 2013; revised February 15, 2014. This work was supported in part by the TM Research & Development under Grant No. EP20120308003.

Tze Cheng Kueh, Shiao Lin Beh, and Yongson Ooi are with the Multimedia University, 75450 Melaka, Malaysia (e-mail: kuehtzecheng@hotmail.com, slbeh@mmu.edu.my, yongson.ooi@mmu.edu.my).

Dirk Rilling is with Baden-Württemberg Cooperative State University Mosbach, 74821 Mosbach, Germany (e-mail: d.rilling@me.com).

DOI: 10.7763/IJIMT.2014.V5.496

II. Experiment Apparatus

The basin used in the experiment is a cylindrical basin with the inlet channel connected to its tangent. The inlet channel is located at the upper part of the basin, uncovered at top. The bottom part of the basin has a hole at the center. The bottom hole is covered by thin aluminum plates with different size of the hole at middle. In the experiment, two aluminum plates with two different size of hole were used, which are 0.02m and 0.025m. The hole discharges the water to another tank under the basin. The water in the tank is then pump up by a pump connecting the tank and the inlet channel of thebasin. The pump model is Leo XHm/5A, centrifugal pumps with maximum flow rate of 650L/minutes. The flow rate of the pump is adjustable from a valve attached to the pipe connecting to the pump. The water is circulated within the system.

Fig. 1 shows the overview of the experiment apparatus, and Fig. 2 shows the sketch and the labeled symbols of the basin parameters. The parameters of the basin are show in Table I.

Fig. 1. Arrangement of apparatus.

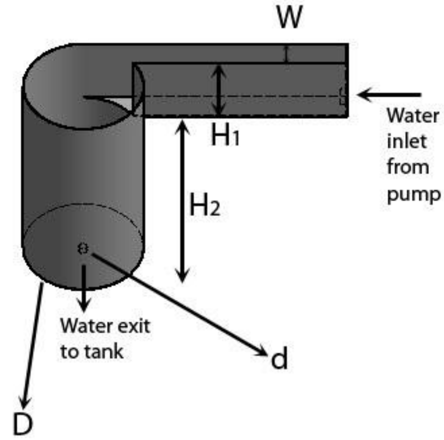


Fig. 2. Sketch of basin and labels of symbols.

The flow rate of the inlet is manipulated from a valve attached to the pipe connecting the pump and basin. However, the accurate adjustment is not available. The flow rate is fixed at a specific rate before the experiment, and the water is let to flow until it reaches steady state. The steady state is determined when the water level remains at the certain height, meaning that the inflow rate and the discharge rate of the water are the same.

TABLE I: Parameter of the Experimental Basin

Parameter	Symbol	Value
Basin diameter	D	0.4m
Basin total height	$H=H_2+H_1$	0.75m
Inlet channel height	H_1	0.25m
Inlet channel width	W	0.09m
Discharge hole diameter	d	0.025m/0.02m
Inlet flow rate	Q	vary

TABLE II: Experimental Water Vortex Height

	Diameter of discharge hole (m)	Discharge rate (kg/s)	Vortex Height (m)
Condition 1	0.020	0.370	0.25
Condition 2	0.025	0.370	0.15
Condition 2b		0.415	0.17
Condition 2c		0.460	0.19
Condition 2d		0.480	0.45
Condition 2e		0.675	0.53

At this state, the discharge rate is measured to determine the inflow rate as well. It is done by collecting the discharged water within 10 seconds, and measure the water volume collected. The height of water vortex is measured from the top of the basin to the vortex highest surface, at the basin wall.

III. Computational Fluid Dynamics: XFlow

A. Introduction to XFlow

The CFD modelling software used in the study was XFlow 2013 Build 90. XFlow uses particle-based and fully Langrangian approach to solve the fluid problems. Different from Eularian approach, it is grid independent on fluid domain. These features allow XFlow to avoid the difficulty of the meshing, which spend most of the effort of users in some other CFD software. In XFlow, users need to set the resolution scales at different part of fluid domain according to the complexity of the flow around the respective parts. Hence, complex geometries, moving parts, and deformable parts are able to be handled in XFlow

XFlow interface contains three major layout i.e., project tree mainly used to set up the model, graphic view, and function viewer used to extract the desired data by plotting graph on certain parameters. The flow model in XFlow divided into single phase and free surface model. Each of them can be divided into external or internal flow.

B. Model Setup

Free surface internal modeling was chosen as the fluid domain in this study. It is bounded by the custom basin shape, as shown in Fig. 2. The gravity effect was set as $9.81 m/s^2$ and initial water level was at 0.5m from the bottom of basin. Fluid used in the model was water, with $998.3 kg/m^3$ density and 0.001Pa•s of dynamic viscosity.

The basin geometry was imported from Autodesk Inventor as a .stp file. Its orientation was reversed so that the surfaces are contact with the water inside the geometry, like a container. By default, the whole boundary conditions of the basin were set as wall. The wall model in XFlow is Wall-Adapting Local Eddy (WALE) model.

There is a separated round surface at the right side of the water channel, as indicated by the word "*water inlet from pump*" in Fig. 2. Its condition was changed from wall to mass flow, with value changed in different case. At the middle of the basin bottom, there is a cylinder hole. The surface of the cylinder was selected, and changed its boundary condition to gauge total pressure outlet. The total pressure is set as 0 Pa and backflow of water was denied.

The simulation time is 200s with every second contains 10 frame to be viewed. The refinement algorithm was selected to be adapt to walls and dynamically adapt to wake. For discharge hole of 0.025m diameter the biggest resolution scale is 0.025m, while it is refined to 0.0125m near the inlet and also for the wake resolution. Since the discharge hole is relatively small, a smaller resolution scale was used near the exit that is 0.00625m. The simulation will be more stable if the resolution scale is the factor of the geometry size. Hence, for discharge hole of 0.02m diameter, largest resolution scare is 0.02m, 0.01m near the inlet and for wake resolution, and 0.005m for resolution scale near the exit.

IV. Results and Analysis

A. Experiment Results

There are two type of diameter of discharge holes used in the experiment. One experiment done on first type of discharge hole, and five different conditions are tested in another. The results of the experiment are shown in Table II.

B. CFD Model 1

TABLE III: INPUT PARAMETERS OF CFD MODEL 1 AND MODEL 2

	Diameter of discharge hole (m)	Inlet flow rate
Model 1	0.020	0.37kg/s
Model 2	0.025	0.46kg/s

There were two sets of models simulated in XFlow. Table III shows the inputs of two models, and Fig. 3 shows XFlow output, that is the overall mass, inlet mass flow rate, and outlet mass flow rate in model 1. Fig. 4 shows the water level of model 1 at different time *T*.

At first, she outlet flow rate is greater than inlet flow rate. Hence the overall mass and water height is decreasing. The water vortex start to form at *T*=50s. The water level continue to drop until the overall mass inside the basin becomes constant, when the outlet mass flow approximate to the inlet mass flow at *T*=150s. At this time, the water height from basin ground is 0.25m.

C. CFD Model 2

Fig. 5 show the overall mass and mass flow rate of the model 2, and Fig. 6 shows the water surface profile of model 2. At first, inlet flow rate is lower than outlet flow rate. The overall mass and water level is decreasing until *T*=57s, then start to rise up. The vortex height at this moment is about 0.18m. From *T*=57s onwards, the water level and vortex height start to increase. However, one thing to take note is that the outlet mass flow rate remain almost constant although the water height is increasing. Until the end of the simulation, the trend of water overall mass is still rising.

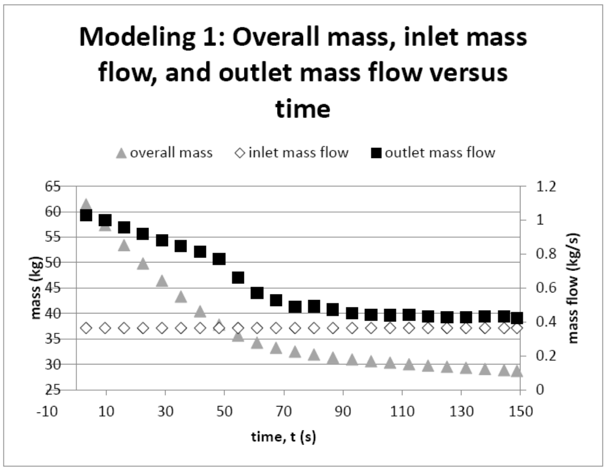


Fig. 3. Total mass, inlet and outlet mass flow versus time: Model 1.

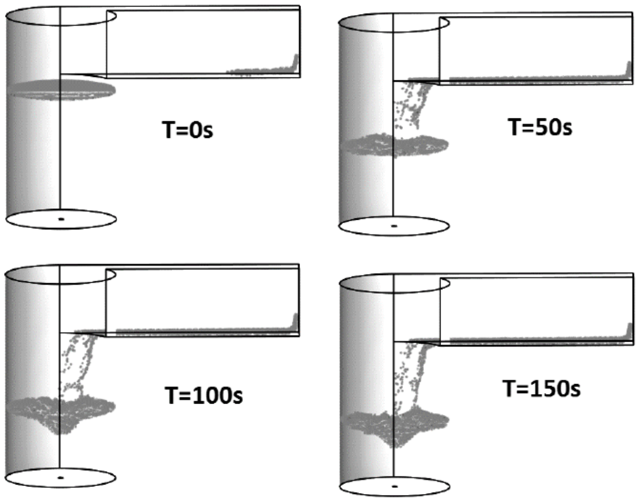


Fig. 4. Water surface profile: Model 1.

D. Comparison and Analysis

From experiment condition 1 and CFD model 1, the input are the same, i.e., discharge hole of 0.02m in diameter and discharge rate of 0.37kg/s. The output of both results are matched. The outlet mass flow will approach the inlet mass flow, and the water vortex reaches a steady height at 0.25m in both cases.

For CFD model 2, it is compared with experiment condition 2c, since they are both having a inlet mass flow of 0.46kg/s. In CFD model 2, when a steady vortex is just start to form, the water height is 0.18m, which is approximate to the result of experiment, which is 0.19m. However, if CFD model 2, the actual discharge rate is only about 0.4kg/s. That is why the vortex height keep increasing in the CFD model 2. Although the vortex height increases, the discharge rate is not increasing as expected.

This can be explained by the strength of the vortex increasing with the increasing height of the vortex. Odgaard [2], Hite and Mih [3] derived an equation to describe the relation between the vortex height and the maximum tangential velocity, shown in (1):

$$H = 1.7\frac{V_m{}^2}{g} \quad (1)$$

where H is vortex height, V_m is maximum tangential velocity, and g is gravity acceleration. When the vortex strength increase, the water motion inside the vortex tends to move in tangential direction instead of axial direction. Hence, the discharge rate should decrease in this case. However, the increasing of the water height also increase the head drop of the water, and the discharge rate at the bottom hole should be increasing. This two effect cancel out each other. Thus the discharge rate remains constant as the vortex height increase.

This effect actually occurs in experiment as well, but at the same inlet flow rate. Table II shows the different variation of experiment condition 2. For the discharge rate increased from 0.37kg/s to 0.46kg/s, the vortex height also increase within an acceptable trend, from 0.15m to 0.19m. However, when the discharge rate reaches to 0.48kg/s and 0.675kg/s, the vortex height jump to another level that is 0.45m and 0.53m. Again, this can be explained by the two effect, i.e., increasing vortex strength and increasing head drop cancelled out each other.

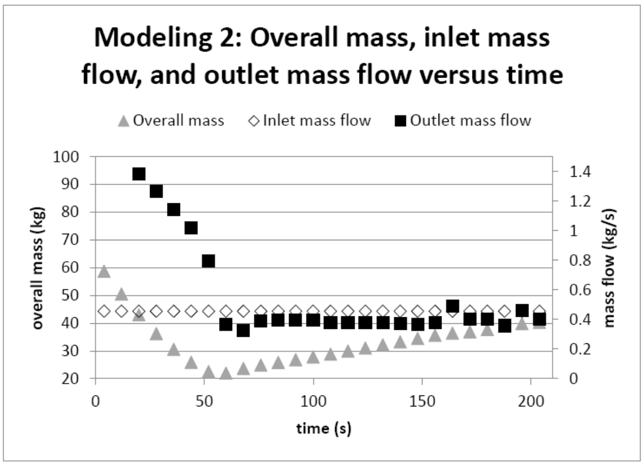


Fig. 5. Total mass, inlet and outlet mass flow versus time: Model 2.

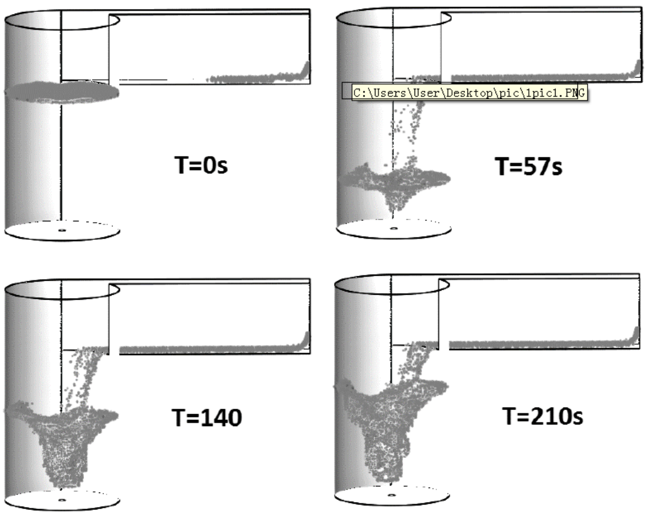


Fig. 6. Water surface profile: Model 2.

For the first three conditions, the inlet flow rate is lower than the outlet flow rate when the vortex just formed. Hence, the vortex height will decrease and remain constant when the outlet flow rate drop to the value of inlet flow rate. This is similar to the CFD model 1. For experiment 2d and 2e, the inlet flow rate in larger than the outlet flow rate when the vortex just formed. Hence, the vortex height rise up, the vortex strength increase, and the outlet flow rate remain almost constant. This explain why the vortex height can be so much different when the inlet flow rate increased from 0.46kg/s to 0.48kg/s. This is similar to the CFD model 2.

V. Conclusion

To design an efficient WVPP, study on self-organized water vortex in a basin is needed. It is time and cost consuming to test every conditions of the water vortex in different geometry. Hence, CFD is a powerful tools that can be used to predict a characteristic of the water vortex.

In this work, a vortex flow is able to be simulated by the XFlow, and the details of it can be extracted from XFlow as well. The outcomes of XFlow in model 1 matches with the experiment setup in condition 1. The final vortex height in model 2 does not match with the experiment condition 2. However, the initial vortex height and the trend that the vortex tend to act when the water level increase are similar in both CFD and experiment condition.

Acknowledgment

The authors acknowledge the financial support provided by TM R&D.

References

[1] W. J. M. Rankine, *Manual of applied mechanic*, London, England: C. Griffen Co., 1858.

[2] A. J. Odgaard, "Free-surface air core vortex," *Journal of Hydraulic Engineering*, vol. 112, no. 7, pp. 610-620, 1986.

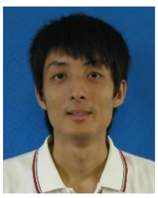
[3] J. E. Hite Jr. and W. C. Mih, "Velocity of Air-Core Vortices at Hydraulic Intakes," *Journal of Hydraulic Enginerring*, vol. 120, no. 3, pp. 284-297, 1994.

[4] C. Yun-liang, W. Chao, Y. Mao, and J. Xiao-ming, "Hydraulic Characteristics of Vertical Vortex at Hydraulic Intakes," *Journal of Hydrodynamics*, vol. 19, no. 2, pp. 143-149, 2007.

[5] J.-P. Torre, D. F. Fletcher, T. Lasuye, and C. Xuered, "CFD Modelling of partially baffled agitated vessels with free surfaces," presented at Fifth International Conference on CFD in the Process Industries, CSIRO, Melbourne, Australia, 2006.

[6] A. Busciglio, F. Grisafi, F. Scargiali, and A. Brucato, "On vortex shape in unbaffled stirred vessels as mesured by digital image analysis," presented at 14TH European Conference on Mixing, Warszawa, 2012.

[7] T. Brown, *More implosion than explosion*, BSRF, Inc., 1986.



Tze Cheng Kueh is from Sarikei, Sarwak, and was born in Malaysia, on March 9,1988. He graduated from bachelor of engineering in mechanical from Multimedia University, Melaka, Malaysia in 2012.

He is currently a research scholar under Research Management Center (RMC) of Multimedia University, Melaka. He worked as an intern in a manufacturing company at Melaka for 16 weeks. His current interest on hydro power plant design specifically on water vortex power plant.

Mr. Kueh currently is not a member of any professional societies.

Shiao Lin Beh obtained bachelor's degree in mechanical engineering from University of Science Malaysia (USM) in 1999, master's degree from the National University of Singapore in 2000 and PhD from Multimedia University (MMU) Malaysia in 2011.

He is currently a senior lecturer in MMU, Melaka campus. He was a research officer in USM before joining MMU, performing numerical modeling on heat transfer in electronic devices. His research interests include

numerical modeling on heat transfer and fluid flow specifically on water vortex.

Dr. Beh is a graduate member of Board of Engineers Malaysia and the Institution of Engineers, Malaysia.

Dirk Rilling was born in Heilbronn/Germany in June 1969, aged 44. He earned at master degree in engineering (Dipl.-Ing.) from the RWTH Aachen University, Germany, in 1995. His field of study was furnace design and heat technologies. His PhD he earn from the same institution in 1999. Topic of his thesis was the determination of annual emissions of houses due to heating.

He worked as a systems engineer for Silicon Graphics Inc. until 2003 until he joined Multimedia University, Melaka, Malaysia. He was a lecturer there for six years with a two years gap where he worked as a head of research and development for Muhlbauer Technologies, Melaka, Malaysia. Serving as a visiting lecturer at the Baden-Württemberg Cooperative State University Mosbach, Germany in 2013 he is an appointed professor for mechanical engineering from January 2014 onwards at the University of Applied Sciences AKAD in Stuttgart, Germany.

He has published several papers about energy optimised design of buildings as well as a book chapter in Sustainable Development Weltanschauung: Beyond Theories into Reality, Prinsip, Malaysia, 2011, about thermal retrofit of buildings. His research interest is in sustainable development, generative design and big data in engineering applications.

Yongson Ooi graduated with mechanical engineering degree from Universiti Sains Malaysia in 2002. He worked as a test equipment engineer in the semiconductor industry in Singapore for 3 years. He completed his master study in 'computer aided geometry design' in 2005 from Universiti Sains Malaysia as well.

He joined KDU College Penang as a lecturer in 2005. He joined Multimedia University as a lecturer in 2007. He has more than 8 years of teaching experience in fluid dynamics, thermodynamics and engineering mathematics. The author current research interest is on 'Computational Fluid Dynamics'. He has completed several research projects including the project in tidal stream energy and flameless combustion funded by MoSTI and MoHE respectively. He has published several research papers for the projects. This paper is the work from the water vortex power plant project funded by TM R&D.

Mr. Ooi is a member of 'Institute of Engineers Malaysia' (IEM).