

Application of multiphase modelling for vortex occurrence in vertical pump intake – a review

M L Samsudin¹, KM Munisamy² and S. K. Thangaraju²

¹Malakoff Power Berhad, Kuala Lumpur, Malaysia, mlutfi.samsudin@malakoff.com.my

²College of Engineering, Universiti Tenaga Nasional, Kajang, Malaysia,
Kannan@uniten.edu.my

Abstract. Vortex formation within pump intake is one of common problems faced for power plant cooling water system. This phenomenon, categorised as surface and sub-surface vortices, can lead to several operational problems and increased maintenance costs. Physical model study was recommended from published guidelines but proved to be time and resource consuming. Hence, the use of Computational Fluid Dynamics (CFD) is an attractive alternative in managing the problem. At the early stage, flow analysis was conducted using single phase simulation and found to find good agreement with the observation from physical model study. With the development of computers, multiphase simulation found further enhancement in obtaining accurate results for representing air entrainment and sub-surface vortices which were earlier not well predicted from the single phase simulation.

The purpose of this paper is to describe the application of multiphase modelling with CFD analysis for investigating vortex formation for a vertically inverted pump intake. In applying multiphase modelling, there ought to be a balance between the acceptable usage for computational time and resources and the degree of accuracy and realism in the results as expected from the analysis.

1. Introduction on Pump Intake Vortex Problem

In many power plants, the cooling water pump system is one of the essential auxiliaries to ensure uninterrupted power dispatch. As reported by [1], two fundamental types of problems for the water intakes are sedimentation and swirling flow problems, occurring at the entrance to or within the pump intake. Sedimentation problems can lead to pump operational difficulties due to it being either conveyed with the diverted flow or deposited in the intake and interferes with the flow. Meanwhile, swirling flow problems, which are commonly related to vortex formation, can lead to swirl and air entrainment at the pump suction bell.

The problem of vortices had been classified into (a) free surface vortices and (b) sub-surface vortices. Surface vortices originate from the free water surface and can have the strength and appearance varying from a slight dimple to a strong circulation or an open funnel or core that would connect the free surface to the pump bellmouth. Floating debris or air can be sucked into the pump column by this strong vortex.

On the other hand, sub-surface vortices are further classified into the floor-, backwall- and sidewall-attached vortices, each referring to the connectivity of the vortices at its one end to the respective boundary walls. Strong subsurface vortex has a relatively strong core and with low



pressure, it can lead to the separation of dissolved air and water to form bubbles within the pump intake flow.

Several difficulties resulted from swirl and air entrainment as highlighted by [2], among others, include operational problems by the intakes of debris, reduced flow rate and head, reduced efficiency, increased vibration and noise and damages incurred on pump critical components. All of these can lead to increased maintenance costs and frequent pump downtime.

The remedies for eliminating these vortices are with proper design of the pump intake and by having adequate submergence. There are several published guidelines such as ANSI/HI 9.8 (1998) [3] and BHRA (1977) [4], to name a few, which provide recommendations for general pump intake design in reducing the vortex formation. ANSI/HI 9.8 also recommends for physical hydraulic model study to be done for pump intakes whose design deviates from the guidelines, having non-uniform or non-symmetrical approach flow, a single pump with flow greater than 2,520 litres per second or a multi-pump station with all pumps running with flow greater than 6,310 litres per second, or where the cost of a model study is 10 times less than the cumulative impact of pump inadequate performance or failure [3].

Model study is restrictive since it is very expensive, site specific, time- and resource-consuming and requires experience from the modellers. Hence, an attractive emerging technique for studying vortex formation is through CFD analysis which is relatively cheaper, quicker and readily available with the commercial codes such as CFX, FLUENT, STAR-CCM+, etc.

The purpose of this paper is to describe the application of multiphase modelling with CFD analysis for investigating vortex formation for a vertically inverted pump intake.

2. CFD Analysis and Multiphase Modelling

The governing equations applicable for the analysis of flow field of pump intake are the continuity equation and the Navier-Stokes equation of motion.

Continuity equation;

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \quad (1)$$

Navier-Stokes equation of motion;

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = \rho f_i - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[(\mu + \mu_t) \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] \quad (2)$$

To match numerical simulation with available computing resources and time, a number of assumptions and simplifications would have to be considered. One of the elements in the CFD analysis is for the fluid flow to be of either single or multiple phases, i.e. liquid water only or liquid water with gas/air/solids.

The differences between multicomponent and multiphase were distinguished. Multicomponent is when the substances are mixed at a microscopic or molecular level and would have a common field of velocity, temperature, etc, for all components in the mixture. Meanwhile, multiphase is when the substances are mixed at macroscopic level and it can be either homogeneous or inhomogeneous. Homogeneous is when common fields of variables would apply for all fractions of the mixture while inhomogeneous is when only common pressure field would apply for all fractions [5].

When a vortex is weak, the air content in water is low and its presence does not exert any significant influence to the flow field. As the vortex's strength increases, the increased air content would lead to a decrease in the pump head, capacity and power consumption together with the change of the air-water mixture density. For simulating such flow, one can apply (a) the Lagrangian approach (or Particle Transport Model), (b) the Eulerian Approach with Particle Model, or (c) the Algebraic Slip Model [5].

On the other hand, when a vortex is strong, the air content in water is high and air bubbles will merge together and form an air core that would connect from the free surface to the pump bellmouth. The air core substantially changes the flow pattern within the intake and exerts influence on the pump performance parameters. Both air and water phases are treated as continuous medium and in simulating this, one can apply (a) the free surface flows model, either homogeneous or inhomogeneous, or (b) the mixture model [5].

The study of vortex formation using multiphase modelling had been done with the homogenous model with clearly defined interface between the phases, by applying the Volume of Fluids (VOF) method. The technique is applied to a fixed Eulerian mesh with objective to track the movement of the interface of two (or more) immiscible fluids. The fluids share the same set of the continuity, momentum and energy equations and the volume fraction of each fluid in each computational cell is tracked over the domain [6].

3. Multiphase Modelling of Single Pump Intake

The modelling of single pump intake for single and two-phase analysis was done almost similarly. For a single phase modelling, 3 blocks of computational domains were constructed consisting of the approach channel, pump lower area and pump outlet/upper area. For a complete two-phase (air and water) modelling, 3 blocks of computational domains were constructed consisting of the free surface upper area (fully air), free surface lower area (fully water) and the intake pipe (fully water). Refer to Figure 1 and 2 for the single pump intake model.

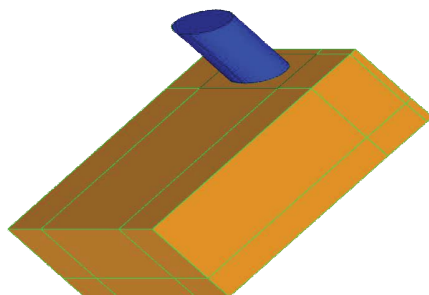


Figure 1: Single pump intake model for single phase modelling.

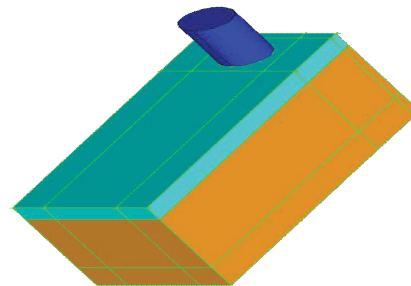


Figure 2: Single pump intake model for two phase modelling.

From [7], the boundary conditions was taken as the total hydraulic head equalled to the assumed water elevation and the submergence level at the inlet while a constant volume flow rate inside the intake pipe at the outlet. Water is considered as monophasic and incompressible with physical models

activated by gravity and turbulence. A third order approximation was used to calculate the momentum advection, while choosing the Split Lagrangian method for the advection of the VOF method.

There are generally 2 approaches used by past researchers to model the air entrainment from the surface vortex occurrence which is the conventional fixed water surface method (C-FWSM) and multiphase free water surface method (M-FWSM). C-FWSM method is economical with lower computational resources, time and cost, but it is only effective with flat and rigid water surface and the formation of vortices tends to be undermined. M-FWSM, on the other hand, is resource- and time-consuming but it is more flexible and accurate. Although C-FWSM needs twice the physical time compared to M-FWSM, to reach periodicity, it is about 5 to 7 times quicker with the same computational settings [8].

For C-FWSM, a free slip boundary is assumed at the water-air interface with particles sharing the same velocity, pressure and temperature field. As such, the same set of basic equations for momentum, energy and mass transport are used for the flow field. For M-FWSM, the simulation of the water surface disruption and distortion can be made without any assumptions on the nature of the fluid interface.

In a study to compare the two methods, the simulation using M-FWSM was able to predict both surface and sub-surface vortices as observed from the experiment whereas C-FWSM was found to be able to predict all except those originated from the back-wall [8].

Volume fraction is an important factor to cause surface vortex appear in the multiphase simulation. A study with steady single phase simulation showed that for a volume fraction of 0.5, i.e. the amount of water is the same as that of air, no vortex appeared. Using volume fraction of 0.95, the vortex becomes visible while increasing the volume fraction to 0.99, the vortex becomes strongly represented [9].

4. Steady and unsteady single phase flow simulation

Most of the earlier research applied steady single phase simulation to study pump intake vortex formation and its strength and position are comparable to the experimental results. As per [10, 11], in modelling the surface vortex formation, using this with $k-\epsilon$ turbulence model, the results were with good convergent, symmetric and stable that agreed well with the experimental results. Meanwhile when used with $k-\omega$ turbulence model, the results were intermittent, asymmetric and unstable and hardly converged.

Steady state simulation had been successfully used to check the effectiveness of anti-vortex device which saw no presence of vortices in the pump intake [8, 12, 13].

5. Steady and unsteady multiphase flow simulation

Steady two-phase flow simulation was applied in studying air entrainment at lowest water level which agreed well with the lab experimental study. As per [14], the simulation was also applied to confirm the effectiveness of the anti-vortex modification by raising the height of the curtain wall, of which the absence of vortex was confirmed.

As per [10, 11], as one of the employed calculation cases, the steady state condition was used as initial condition for unsteady flow simulation which gave better convergence results. The occurrence of air entrainment was found to increase with increasing mass flow rate. It was easier to be visualised by using volume of fluid of air in defined cut planes but the great difficulty was to locate these planes due to unsteady position of the air core. The strength of the predicted vortices was not equal and changes with time. It seemed to follow a specific frequency which is based on mean shear velocity and pipe diameter.

The basic flow characteristic of water within the pump intake is highly unstable, unsteady and intermittent, independent of the turbulence model used [10, 11]. In most cases, a fully unsteady state simulation is not always needed for analysis and troubleshooting as the steady state simulation can be sufficient in explaining the phenomenon for vortex formation with significantly lower computational time.

6. Multiphase Modelling of the Sub-surface Vortex

As per [15], VOF method can be applied to predict the incidence of sub-surface vortices if being used with appropriate phase-change model, e.g. cavitation.

A CFD study was made of a practical model and model built as per Turbomachinery Society of Japan (TSJ) Standard S002:2005, as per [9]. Applying single phase simulation for unsteady flow for sub-surface vortex formation of single pump intake showed that the floor-attached vortex was observed just until 7.5 seconds and then it was not represented with low vorticity. When two-phase simulation was applied, the development of the vortex always changes and not fixed, in agreement with the observation from the experimental study. It was recommended that the unsteady multiphase modelling is best applied to study the sub-surface vortices.

7. Multiphase Modelling of the Sediment

The use of multiphase, or perhaps multicomponent, modelling for pump intakes can further be extended to simulating other unexplored real-life factors influencing the vortex formation. As considered and assumed at design stage, the intake fluid flow is taken as single phase in a sense that it is free of air as well as solid particles. Although there are several design fixtures that can be taken in minimising course and fine sediment transport, as highlighted in [16], this phenomenon does occur following years of operations, especially for the case of offshore pumping stations with seawater intakes.

The sediments would have several influences to the flow field within the pump intake. Course sediment would settle to the bottom, accumulate and solidify under hydrostatic pressure over time to permanently change the floor profile. Excessively low floor clearance and converging floor profile can increase the mass flow rate towards the pump bellmouth, thereby inducing vortex formation. On the other hand, fine sediment would flow with the water intake and carried along, influencing the density and energy of the moving fluid.

As per [17], an experimental study was conducted to study the effect of silt concentration in pumping water, using samples of clay composed of very thin particles and containing approximately 15% of fine sand. It was found that the NPSH curves are not influenced by the concentration at low flow rate. At high flow rate, the curves of silt mixture appeared higher than those of water only. The cavitation accelerates with the silt concentration, being only sensitive to the viscous flow and not to the gravitational forces.

The use of Eulerian model is recommended for sedimentation analysis [6].

8. Conclusion

Multiphase modelling for flow field analysis within pump intake is becoming key consideration with CFD. It has the potential to produce accurate results in determining the formation of both surface and subsurface vortices, especially for the case of air core and entrainment phenomenon. There ought to be a balance between the high demand for computational time and resources and the degree of accuracy and realism in the results as expected from the analysis.

9. References:

- [1] Melville B W, Etterna R and Nakato T 1994 *Review of Flow Problems at Water Intake*

- Pump Sumps* EPRI, Palo Alto CA Electric Power Research Institute TR-103474
- [2] Knauss J 1987 *Swirling Flow Problems at Intakes (IAHR Hydraulic Structures Design Manual)* (New York: Taylor & Francis).
 - [3] ANSI/HI 9.8 1998 *American National Standard for Pump Intake Design* American National Standards Institute Inc.
 - [4] Prosser M 1977 *The Hydraulic Design of Pump Sumps and Intake* British Hydromechanics Research Association/Construction Industry Research and Information Association.
 - [5] Kochevsky A N 2005 Capabilities of Numerical Simulation of Multiphase Flows in Centrifugal Pumps using Modern CFD Software <http://arxiv.org/abs/physics/0509193>
 - [6] ANSYS FLUENT Theory Guide Rel 14.0 2011.
 - [7] Lucino C, Liscia S and Duro G 2010 Vortex Detection in Pump Sumps by Means of CFD XXIV Latin American Congress on Hydraulics, Punta Del Este, Uruguay, Nov 2010
 - [8] Kurger S, Bouziad Y A and Maurer W 2009 Pump Sump CFD for Vertical Pump Design *Proc of the ASME 2009 Fluids Eng Div Summer Mtg, FEDSM2009, Vail, Colorado, US Aug 2-6, 2009*
 - [9] Park Y K, Li K M, Chi Y H and Lee Y W 2011 A Study on the Generation of Vortices around a Sump Intake with Multiphase Flow 11th Asian Int Conf on Fluid Machinery and the 3rd Fluid Power Tech Exh, IIT Madras, Chennai, India Nov 21-23, 2011
 - [10] Bayeul-Laine A C, Simonet S, Bois G and Issa A 2012 Two-Phase Numerical Study of the Flow Field Formed in Water Pump Sump: Influence of Air Entrainment 26th IAHR Symp on Hydraulic Machinery and Systems; IOP Conf Series: Earth and Env Science **15** (2012) 022007
 - [11] Bayeul-Laine A C, Simonet S, Issa A and Bois G 2013 The Importance of the Knowledge of Flow Sream in Water Sump Pump *Congres Francais de Mechanique (21;2013; Bordeaux), France, 2013-08-26*
 - [12] Kim C G, Choi Y D, Choi J W and Lee Y H 2012 A Study on the Effectiveness of an Anti Vortex Device in the Sump Model by Experiment and CFD 26th IAHR Symp on Hydraulic Machinery and Systems, IOP Conf Series: Earth and Env Sciences **15** (2012) 072004
 - [13] Cheng L and Liu C 2012 Research on Vortex Eliminating of the Sump of Cooling Pump by CFD *Proc of the 2012 20th Int Confon Nuclear Eng collocated with ASME 2012 Power Conf (ICONE20-POWER2012) Anaheim, California, USA, Jul 30 – Aug 3, 2012*
 - [14] Shukla N S, Kshirsagar J T 2008 Numerical Prediction of Air Entrainment in Pump Intakes *Proc of the 24th Int Pump Users Symp (Texas A&M University, USA) pp 29-33*
 - [15] Zhao L J, Nohmi M 2012 Numerical Simulation of Free Water Surface in Pump Intake 26th IAHR Symp on Hydraulic Machinery and Systems, IOP Conf Series: Earth and Env Science **15** (2012) 052035
 - [16] Zheng Y, Alsaffar M A 2004 Water Intake Sediment Problems in Thermal Power Plants *Joint Conf on Water Resources Engineering and Water Resources Planning and Management (Minnesota, USA) Jul 30 – Aug 2, 2000*
 - [17] Ladouni A and Nemdili A 2005 Experimental Investigation of the Effect of Solid-Mixture on the Cavitation Characeristic of a Centrifugal Pump *Fluid Structure Interaction and Moving Boundary Problems, WIT Transactions on the Built Environment* **84** pp 81-87