Wiroon So-in
Jutarat Kurujareon
Lt.Col. Anotai Suksangpanomrung

Jointed Post Graduated Program (SWU-CRMA)
Department of Mechanical Engineering,
Faculty of Engineering Srinakharinwirot University

The Study of Vortex Prevention in Sump with Vortex Breaker by Computational Fluid Dynamics

Abstract

The flow characteristic in sump with suction pipe was numerically studied utilizing Computational Fluid Dynamics (CFD) technique based on a finite volume method (FVM). The Reynolds-Average Navier-Stokes (RANS) equation with two-layer k-e model was solved unsteadily through out the numerical model. It was found that our numerical results agreed well with the experimental results conducted by (4).

Moreover, in our studied, a simple geometry vortex breaker was added at the bottom of sump for the purpose of preventing the vortex. And it was observed that only floor surface plane that vortex was weakening, the other three planes along the sump walls gave a greater strength with additional small vortexes surrounding the breaker.

Keywords: sump, submerged vortex, vortex breaker, CFD

1. Introduction

In the designing of water sump process, one of the most important things that must bring into the consideration is the occurrence of the cavitations since they are directly harmful to the pumping system. They reduce the efficiency and the operating life of the pump by hydraulically impacting to the pump impeller and casing (1), and in addition, the cavitations also cause the structure vibration and the noise through out the building.

Cavitations could be classified into two types as the way they were inception, 1) Free surface vortex and 2) Submerged vortex. The latter is the major concerns since they are more complicated in the occurrence and prevention. Thus vortex breakers are often used to prevent both type of vortex. Due to the various applications and actual site conditions, many types (2) of vortex breakers are provided. However, there are not much studies of the flow characteristics in such applications. Only a few studies of the following researches were conducted.

Constantinescu (3) developed the numerical model to simulate three dimensional flow field in the pump intake bay. The model was solved by RANS with the two-layer k-ε model in curvilinear coordinate. The near wall flow was also taken into account to this turbulent model which is important for prediction of the wall attached submerged vortex. The numerical model was approximately 550,000 of grid points and was tasted for the grid dependency with lower resolution to 220,000 with no difference more than 4%. The calculation also continued on Reynold's number of 20,000 and 60,000 on the coarse and fine grid, respectively.

Rajendran (4) conducted the laboratory sump to validate his numerical model. The flow properties in the sump were measured by PIV and the result was compared to the numerical model calculated by using CFD. The model of the Reynolds-Average Navier-Stokes (RANS) equation was solved with two-layer k-ε model at steady state condition. The results came out agree well with experimental one. This experimental model of the sump without vortex breaker will also be used as a validation for our investigation before adding the vortex breaker in the sump.

Constantinescu (5) had compared the two type of turbulent models for their roles and the influence of wall roughness on the prediction of location, size, and strength of different type of vortexes. The k-ε and k-ω model were used and their result were similar in shape and size but gave a different in strength and location which depended on the near-wall flow treatment. The roughness of the wall was weakening the strength of vortices which may be use as one of vortex suppression.

Ansar (6) also conducted the laboratory model with single and dual pump and compared the results with the numerical model. The cross flow and no cross flow case were taken into account in both experimental and numerical model.

The goal of our research was to study the flow pattern and phenomenon such as the change of streamline, the strength of each vortex, and quantity of vortex before and after the breaker was installed.

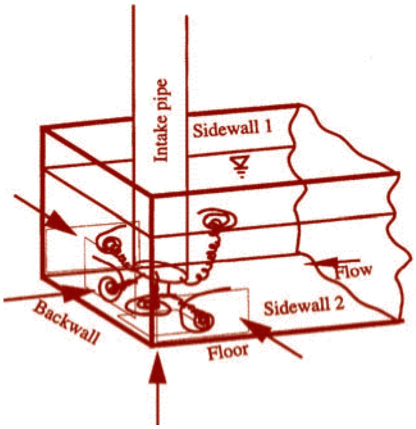

Figure 1 Vortice in the sump (4)

Therefore, there was unnecessary to use the complicated shape and installation location of the breaker.

2. Experimental and Numerical Models

As previously mentioned, the model of Rajendran (4) was use to validated the numerical model, the sump is 308 mm. in width, 1,120 mm. in length, and 211.2 mm. in height, the suction pipe is 88 mm. in diameter and was placed eccentric to the sump to make the imbalance of the incoming flow. Inlet velocity is 0.003 m^3/s and placed at the far end of the sump wall. The Reynolds based on pipe diameter and the average inlet velocity in pipe was 45,000 and the Frude (F) and the Weber (W) Number were 0.55 and 325, respectively. This model could produce one-free surface and four-submerged vortex along the walls bounded the suction pipe as shown in Figure 1

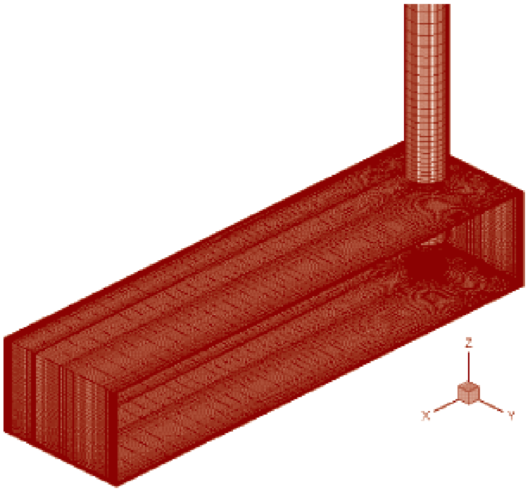

Figure 2 Computational Grid

but since the free surface vortex was not much complicated in prevention, this research was focused only the submerged one, therefore, the comparison of free surface vortices was not shown here.

The numerical were carried out on a structured grid of 578,000 points as shown in Figure 2. RANS and RNG k-ε Model (Enhance Near-Wall Treatment) were solved in 3D with implicit and unsteady condition.

Finally, a vortex breaker was added into the model of sump pump at the bottom of the sump as shown in Figure 3. The flow condition used for this model was the same as the model of the previous one.

Breaker was rectangular with the same width as sump. The height of 70.4 mm was used (Which is exactly equal to the gap between sump bottom and the inlet of suction pipe). The thickness of the breaker was assumed to be infinity thin.

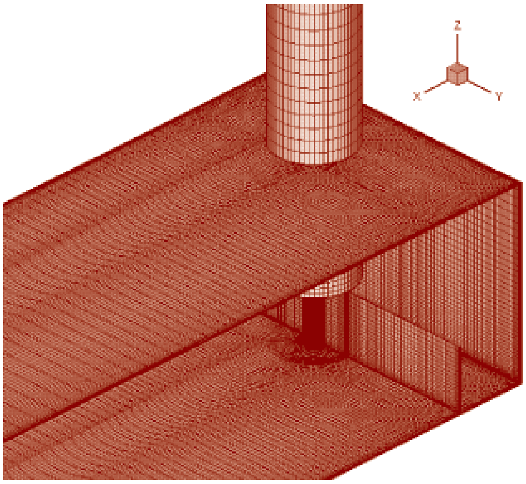

Figure 3 Computational Grid showing Vortex Breaker on the Sump Bottom

The model also was tested for the Grid dependency with a higher and lower resolution of the cell. The results were compared for the location, strength, and the size of vortex.

3. Numerical Results

The numerical results are presented by cutting the plane at some distance parallel to each sump wall and denoted as 1) Floor Surface, 2) Backwall Surface, 3) Sidewall 1 Surface, and 4) Sidewall 2 Surface, the distance from the wall, respectively, were;

- Floor Surface = 0.25d from Floor
- Backwall Surface = 0.23d from backwall
- Sidewall 1 Surface = 0.25d from sidewall 2
- Sidewall 2 Surface = 0.15d from sidewall 1

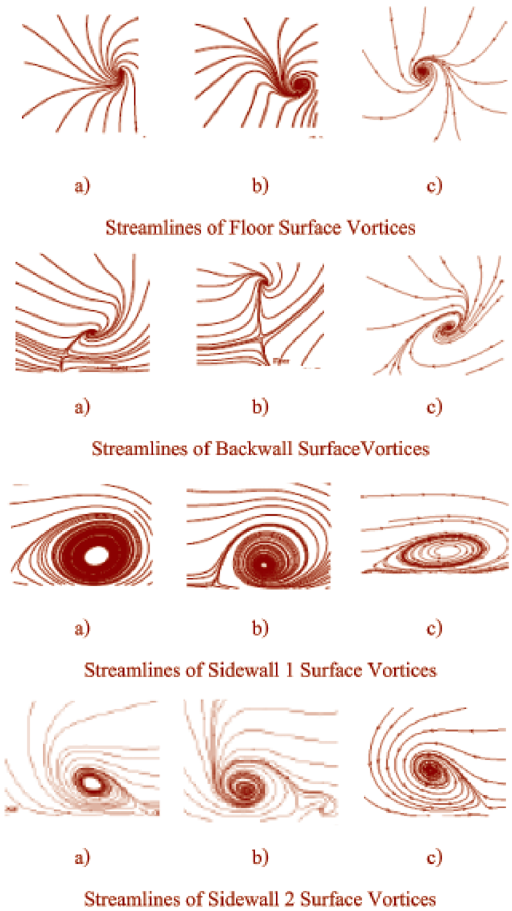

Figure 4 Streamline comparison; a) Numerical Model of referenced paper, b) Experimental model of referenced paper, c) Numerical model of this paper

3.1 Comparison with Referenced Paper

The streamlines are shown in Figure 4 comparing between the experimental and the numerical model. Circulation directions are not different but the location is still in doubt since the referenced paper did not exactly show the location of each vortex at each plane of sump wall.

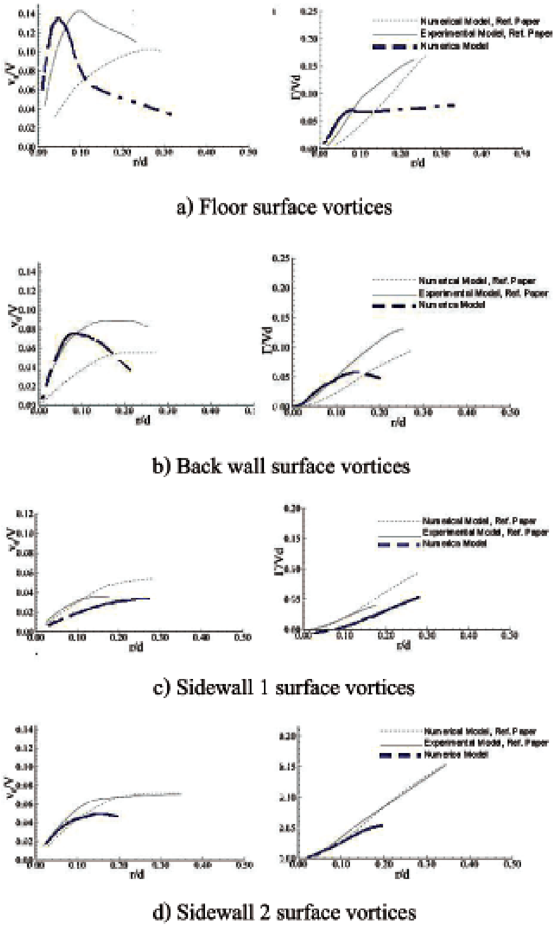

Figure 5 Tangential and circulation comparison

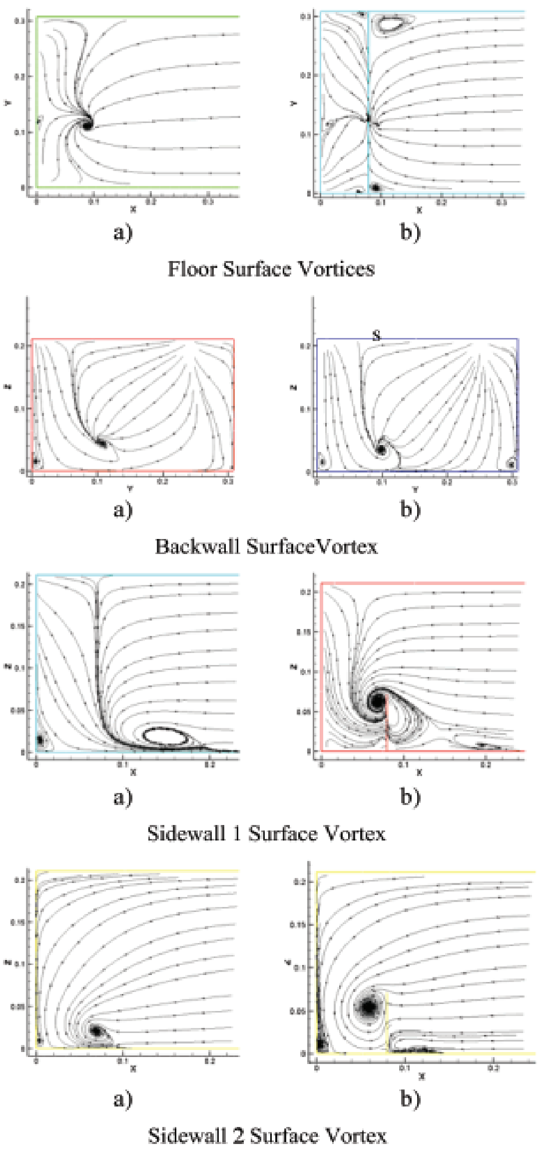

Figure 6 Streamline comparison; a) Without breaker, b) With breaker

As the streamline only do show the graphical comparison, hence, the numerical points of view are required. Figure 5 illustrates the results of calculation compared with the referenced paper, the calculation are presented in the dimensionless parameter as r/d, V_θ/V, and Γ/Vd

The numerical results of the overall data yields the same trend as experimental results. However, the calculation results are closed only at the small radius of vortices. At larger radius, the tangential velocity and the circulation are drop steeply but there are sufficient for this study.

3.2 Numerical Results of Sump with Vortex Breaker

After the vortex breaker has been added to the sump, there are some changes of flow pattern especially at the plane of floor surface, sidewall 1 surface, and sidewall 2 surface. This caused by the change of sump geometry with the additional breaker. However at the back wall surface, there are not much changed of the flow pattern, as shown in Figure 6.

At the floor surface plane, there are 4 additional vortices observed close to the breaker and the sump wall. However, the strength of these vortices is not much grater. While the vortex at the same location where breaker has not been install is weaker as shown in Figure 7.

At sidewall 1 and 2, breaker produces additional vortex in front of the breaker. It is shown in Figure 7 that the strength of vortex at the same locations where the breaker has not been placed is stronger. This indicates that such type of breaker is not applicable to prevent vortex in these two planes.

At the back wall plane, although the flow pattern does not change too much, but the breaker gives a greater strength of vortex as occurred at sidewall 1 and sidewall 2.

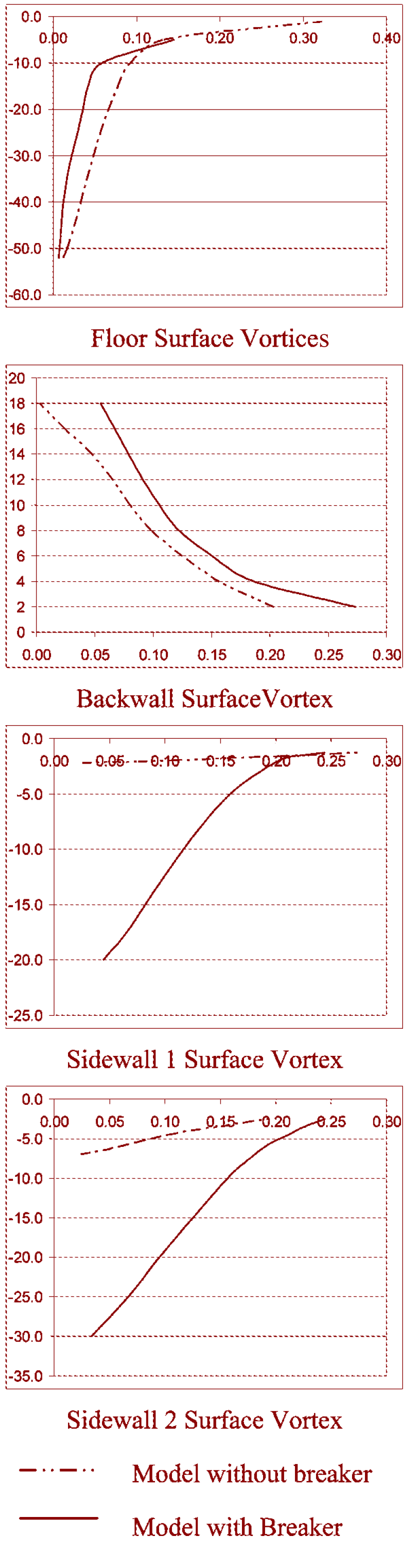

Figure 7 Plotting of Vorticity against the radius (r/d)

4. Conclusion

The calculation results revealed that numerical model gave the deviated flow data from the experimentation. Nevertheless, they were in the same trend and direction especially at the small radius of vortices which the calculation was significantly close to the experimental. This is a cheap and fast way in designing of sump which will give the designers a preliminary conclusion before the real sump with true or smaller scale constructing.

The breaker that added in the sump did not help in prevention for all of the vortexes. On the other hand, the small strength of vortexes were generated around the breaker at some plane. These vortexes, with the flow conditions such as inlet velocity is not change, may not be accounted if they are not creating the air core.

To prevent the vortex in the sump, it requires more than just a simple geometry of vortex breaker since the phenomenon of submerged vortex are too complicated and many times the site condition are totally different from what had been specify in the any handbooks. Therefore, a careful calculation must be taken since it directly effected on the cost and construction time.

The simulation results were given only for the vortex breaker that had a fixed dimension in length, width, and height. As previously mentioned, the vortex at some planes such as backwall and sidewall could not be eliminated. On the other hand, there was stronger vortex strength occurred in this vicinity. It is interesting that such parameters have an influence on the vortex prevention or not. Hence it might be the subjects for the next stage of this research area.

References

(1) Takahida Nagahara, Toshiyuki Sato, Tomoyoshi Okamura, "Effect of the submerged Vortex Cavitation Occurred in Pump Suction Intake on Hydraulic Forces of Mixed Flow Pump Impeller", (Online), Available: http://cav2001.library.caltech.edu/358/00/b8_006_nagahara.pdf#search=%22Effect%20of%20the%20submerged%20Vortex%20Cavitation%20Occurred%20in%20Pump%20Suction%20Intake%20on%20Hydraulic%20Forces%20of%20Mixed%20Flow%20Pump%20Impeller%22. Retrieved December 15, 2004.

(2) John, L. Dicmas, Vertical Turbine, Mixed Flow, and Propeller Pump, McGraw-Hill Book Company, 1987, pp. 177-211.

(3) G. S. Constantinescu, V. C. Patel, "Numerical Model for Simulation of Pump Intake Flow and Vortices", Journal of Hydraulic Engineering, February 1998, pp. 123-134.

(4) V. P. Rajendran, S. G. Constantinescu, and V. C. Patel, "Experimental Validation of Numerical Model of Flow in Pump-Intake Bays", Journal of Hydraulic Engineering, November 1999, pp. 1119-1125.

(5) G. S. Constantinescu, V. C. Patel, "Role of Turbulence Model in Prediction of Pump-Bay Vortices", Journal of Hydraulic Engineering, May 2000, pp. 387-391.

(6) Matahel Ansar, Tatsuaki Nakato, George Constantinescu, "Numerical Simulation of Inviscid Three-Dimensional Flows at Single and Dual Pump Intakes", Journal of Hydraulic Reaserch, Vol. 40, 2002, no. 4, pp. 461-470.

Notation

The following symbols are used in this paper:

r = radial distance from vortex center (m)

d = Suction pipe diameter (m)

V_θ = tangential velocity (m/s)

V = average velocity in intake pipe (m/s)

Γ = circulation (m^2/s)