

## Computation of Critical Submergence Depth to Avoid Surface Vortices at Vertical Pumps Intakes

Ali Fakhri Kadhim \*

College of Engineering - University of Baghdad  
Baghdad, Iraq  
E-mail: a.kadhim1310@coeng.uobaghdad.edu.iq

Asst. Prof. Dr. Hayder A. Al Thamiry

College of Engineering - University of Baghdad  
Baghdad, Iraq  
E-mail: hy\_hyder@coeng.uobaghdad.edu.iq

### ABSTRACT

The pumping station became widely used in many fields. Free surface vortices at intakes of pumps are not favorable. It may cause noise, excessive vibration, damage to the pumping structure, reduction in efficiency and flow for hydro-turbines, etc. One of the important problems encountered during the pump intake design is the depth of submergence and other design parameters to avoid strong free-surface vortices formation. This study aims to compute the critical submergence depth with some geometrical and hydraulic limitations by using Computational Fluid Dynamic (CFD) package. The mathematical model was validated with a laboratory model that had been conducted. The model of three intake pipes was investigated under five different submergence depth (S), three different spaces between intake pipes (b), and five different suction velocities (v). The results showed the best operation cases when the space between intake pipes (b) equal to 4D, the submergence depth of water is equal or greater than 1.25 from the bell mouth diameter of intake pipe (D), and the suction velocity less than 2 m/s. The worst case was when the space between the suction pipe (b) was (2D), in this case, the vortex appeared at submergence depth ( $S/D = 2$ ) with suction velocity 3 m/s.

**Keywords:** Critical submergence, CFD, intake pipe, surface vortices.

### حسابات العمق الغاطس الحرج لتجنب الدوامات السطحية عند المآخذ العمودية للمضخات

علي فخري كاظم  
جامعة بغداد/ كلية الهندسة

أستاذ مساعد الدكتور حيدر عبد الأمير الثامري  
جامعة بغداد/ كلية الهندسة

### الخلاصة

أصبح استخدام محطات الضخ واسعا في العديد من المجالات. تعد الدوامات السطحية التي تحدث عند مآخذ محطات الضخ ظاهرة غير مرغوب بها. هذه الظاهرة ممكن ان تسبب الضوضاء، زيادة في الاهتزازات، ضرر لهيكل الضخ اضافة الى تقليل الكفاءة والتصريف للتوربينات المائية، الخ. من أهم المشاكل التي تؤخذ بنظر الاعتبار عند تصميم محطات الضخ هي تصميم

\*Corresponding author

Peer review under the responsibility of University of Baghdad.

<https://doi.org/10.31026/j.eng.2020.08.05>

2520-3339 © 2019 University of Baghdad. Production and hosting by Journal of Engineering.

This is an open access article under the CC BY4 license <http://creativecommons.org/licenses/by/4.0/>.

Article received: 3/1/2020

Article accepted:11/3/2020

Article published:1/8/2020



العمق الغاطس والعوامل الاخرى لتجنب حدوث دوامات سطحية قوية. هذه الدراسة تهدف الى حساب العمق الغاطس الحرج مع بعض المحددات الهندسية والهيدروليكية وباستخدام حسابات حركة الموائع (CFD). تم التحقق من النموذج الرياضي عن طريق نموذج مختبري سبق وأن تم تنفيذه. النموذج الذي تم دراسته يتألف من ثلاثة مأخذ وتمت الدراسة ضمن خمسة اعماق غاطسة (S)، وخمسة سرع سحب مختلفة (v) وخمس مسافات مختلفة بين انابيب السحب (b). بيّنت النتائج إن افضل الحالات التشغيلية عندما يكون العمق الغاطس اكبر أو مساوي لـ(1.25) من قطر مدخل أنبوب السحب، والمسافة بين أنابيب السحب مساوية لأربعة أضعاف قطر مدخل الأنبوب وسرعة السحب لا تزيد عن 2 م/ثا. اما أسوء حالة ظهرت عندما كانت المسافة بين أنابيب السحب مساوية لضعف قطر مدخل انبوب السحب، في هذه الحالة فإن الدوامات استمرت بالظهور عندما كان العمق الغاطس مساو لضعف قطر مدخل انبوب السحب وسرعة السحب مساوية لـ(3 م/ثا).  
الكلمات الرئيسية: العمق الغاطس، إنبوب مأخذ، CFD، دوامات سطحية.

## 1. INTRODUCTION

Pumps are widely used in providing us with drinking water, industrial water supply, agriculture irrigation, city drainage and drawing water from rivers reservoirs, etc. The design of sump tanks and pump intake is usually based on standard design manuals, and empirical formulae got from tests of the experimental model previously prepared. Many of the problems which may happen at pump intakes are a result of free-surface vortices. Surface vortex is often a highly turbulent flow phenomenon which may happen due to non-uniform flow and the residual angular momentum in the flow at the intake. However, surface vortices are a severe problem which likely to avoid. Free surface vortices have been found that it was the primary problems that cause damage to the pumping system, flow reductions, vibrations, etc.

Most of the researches that focused on free-surface vortices phenomena had performed laboratory hydraulic models for a given hydroelectric plant, pumping station, or spillway intake to determine whether vortices will happen. Some researches had been performed studies for the submergence depth requirements to avoid free surface vortices formation. This study is aimed to simulate the free surface vortex and derive formulae to compute the critical submergence depth for pump intake based on numerical CFD model code ANSYS Fluent. The CFD is a computer-based software. It helps the researchers and designers in simulating the cases of fluid flow, heat transfer, and other characteristics associated with physical processes. It is solving the equations of fluid flow in the region of study, with the required boundary conditions for this case of study.

The review of literature that carried out to avoid surface vortices showed that **(Werth and Frizzell, 2009)** studied some experimental hydraulic models of vertical turbine pump sumps. They used these models to evaluate the existing empirical equations to help the designers in determining the minimum submergence required and prevent the formation of a surface vortex. They carried out ten different un-distorted scaled hydraulic model of vertical turbine cooling-water pump intakes. The intakes were conducted according to the standards of Hydraulic Institute **(HI)** to assess hydraulic performance as well as the required submergence to prevent dye-core vortex activity. They are based on evaluating the equation by **(Gulliver and Rindels, 1987)** for vertically flowing intakes, and they did a refinement for this equation, and the proposed equation takes advantage of the relatively consistent and confirmed intake geometry in the vicinity of the pumps for designing with standards.

**(Kleynhans, 2012)** worked on an experimental hydraulic model to investigate the critical submergence for pump intake. He designed hydraulic experimental model with a scale of 1:10, and he used it in computing the submergence required to avoid vortices with a suction velocity at the bell mouth between (0.9 - 2.4 m/s), and three values of clearance for suction bells above the floor placed at (0.5D, 1.0D and 1.5D). He carried out tests for four suction bell configurations, being the well-liked suction bell arrangement in terms of the lowest required submergence depths. He concluded that critical submergence should be based on the surface vortices Type 2, which is called surface dimples according to the American National Standard for pump intake **(ANSI/HI 9.8)**, because the vortices Type 3 (coherent dye core) is a difficult associated with identifying in the experimental hydraulic model. The results showed that the velocity in the bell inlet should be



specified at 1.5 m/s because the water surface was being turbulent in the experimental model when the velocity at the bell inlet became above 1.8 m/s.

**(Borghei and Kabiri-Samani, 2013)** studied the influence of using an anti-vortex plate on air-entraining by surface vortices. They conducted a set of experiments to reduce the influence of vortices and air entering at vertical pipe intakes. They used a rectangular anti-vortex plate. They concluded that the use of anti-vortex plates led to decrease water depth, moreover, reduction in intensity, and duration of swirls. They showed that the use of large plates led to observe a weak air-core vortex with floating trash, a dye core vortices, and small dimples when the use of small plates detected a full air-core vortex which pulled air bubbles into the intake.

**(Zhao, 2014)** carried out the CFD model for a single suction pipe to study the flow properties in the rectangular sump and analyze the mixed flow that happened. He used constant discharge and submergence depth. He studied a numerical model based on commercial software ANSYS CFX 13. The model designed with flow rate was 88.2 m<sup>3</sup>/hr, distance between inlet bell mouth and the floor was 100 mm, bell mouth diameter is 270.8 mm, suction pipe diameter was 122 mm, and the submergence depth set as 510 mm. The numerical model was carried with the turbulence model (SST) to visualize the occurrence of the surface and submerged vortex. He used a curtain wall installed in the surface and square bar placed across the channel to suppress the surface vortices. The result showed that the use of a curtain wall was effective in preventing the surface vortices, but used the square bar installed across the channel failed to suppress the free surface vortex.

**(Athab, et al., 2017)** carried out an experimental model of three pumps to study the setting of the multiple vertical entrance suction pipes in the stormwater pumping station. They analyzed six factors that may affect vortices formation in pump sump, the velocity of inlet suction pipe ( $v$ ), the submergence depth ( $s$ ), the space between the suction pipes centerline ( $b$ ), the distance of the clearance between the floor and the suction pipes ( $z$ ), the effect of the distance to the back wall ( $a$ ) and the effect of adding baffles between the suction pipes. They conducted 1350 runs to conclude the best hydraulic conditions of the flow in the sump. They measured the diameter of vortices that were formed and the height of it. The results showed that the formation of free surface and sub-surface vortices were mainly affected by flow velocity, followed by pipe spacing, back wall distance, and submergence, and to less influenced by the clearance to the floor ( $z$ ). And they introduced formula to determine the critical submergence as a relation with geometrical and hydraulic parameters.

**(Guo, et al., 2019)** studied the influence of suction pipe intake condition on the appearance of the vortex in pump sump. They worked on an experimental model of pump sump that had been designed and executed with a single pipe to validate the happening of the vortex. In this study, the researchers recorded the value of flow rates, which gave critical submergence. They found that the relationship between discharge and water level was approximately linear. They concluded that the velocity of the vortex core and water swirl is high, and the pressure in the region of the core is lower than in other regions.

This study aimed to apply the CFD code in studying the influence of using three suction pipes and the effect of the distances between these pipes on computing the critical submergence depth.

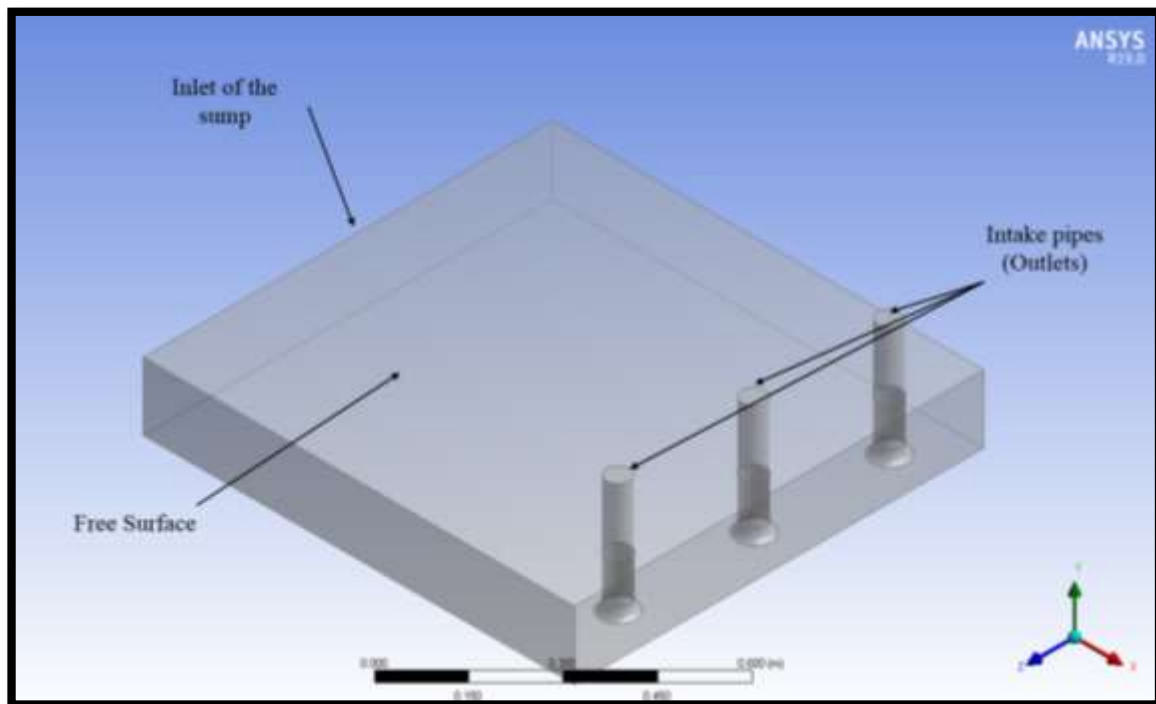
## 2. DESCRIPTION OF THE MATHEMATICAL SIMULATION OF PUMP SUMP AND INTAKES.

Over the past several years, the experimental model was used to define special problems in pump sump and intake and try to solve them. The experimental model is usually expensive, time-consuming, and required experience from the modeler. With the recent development of using commercial (CFD) software, a numerical solution is becoming the best way to address this type of hydraulic modeling. The commercial CFD package of Fluent was used to study and analyze water flow through the pump sump and intakes and specify critical submergence. The process of performing a CFD analysis generally includes the definition for the geometry of the model, mesh generation, the definition of the materials, and using the convenient boundary conditions, solver, and post-processing for the results. The governing equations of flow are solving in the solver process.

## 2.1 Model Generation and Meshing

**Fig. 1** shows a general view of the geometry of pump sump and intakes. The geometries of 3D models for the flow domain of pump sump and intakes pipes were generated by using SOLIDWORKS 2018 software. SOLIDWORKS software used for drawings of the geometric details and helps to sketch out ideas and experiments with exact dimensions.

For validation of the CFD model with the experimental results, some models were drawing depending on the experimental model carried out by (Athab, et al., 2017). Each model had details of dimensions according to its case. The geometries imported and used in ANSYS Fluent code for simulation.

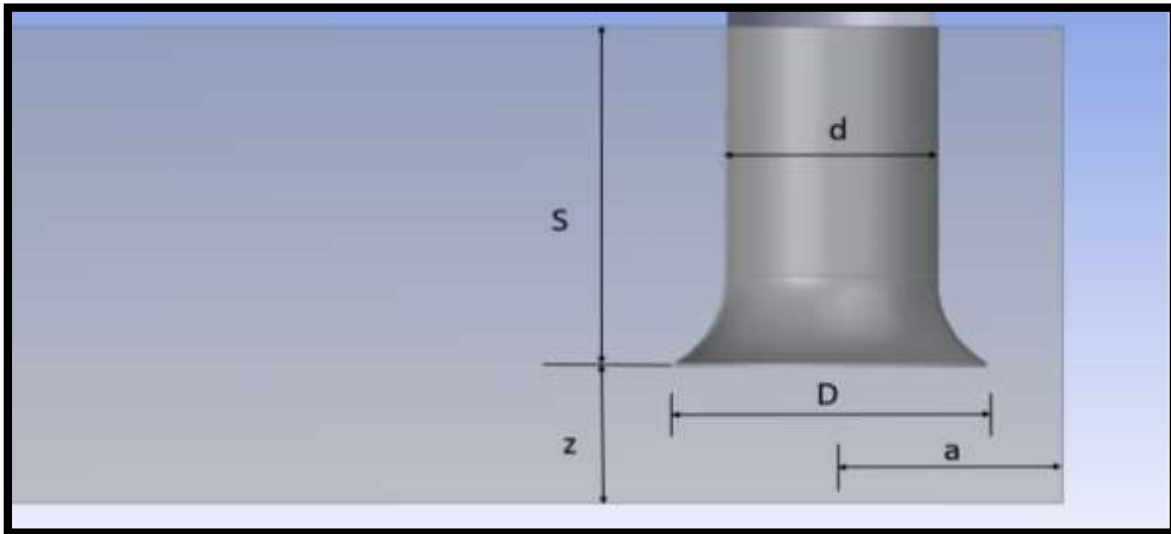


**Figure 1.** General view of the geometry of pump sump and intakes.

After validating the (CFD) model, geometries are drawn to study the effect of the space between the suction pipes of three pumps on the occurrence of the surface vortex to specify the critical submergence depth. The mathematical models had tanks with constant length and width (970 mm) (920 mm), respectively. The diameter of the intake suction pipes ( $d$ ) was (50.8 mm), and the bell mouth ( $D$ ) was (76.2 mm). The distance from the intake suction pipe center to the back wall was placed constant ( $0.75D$ ), and the clearance of the suction pipe to the floor was set ( $0.5D$ ) according to American National Standards for pumping intake design (ASNI/HI 9.8.). **Fig.2** shows the details of the geometry of models. The other parameters were set as in **Table 1**, below.

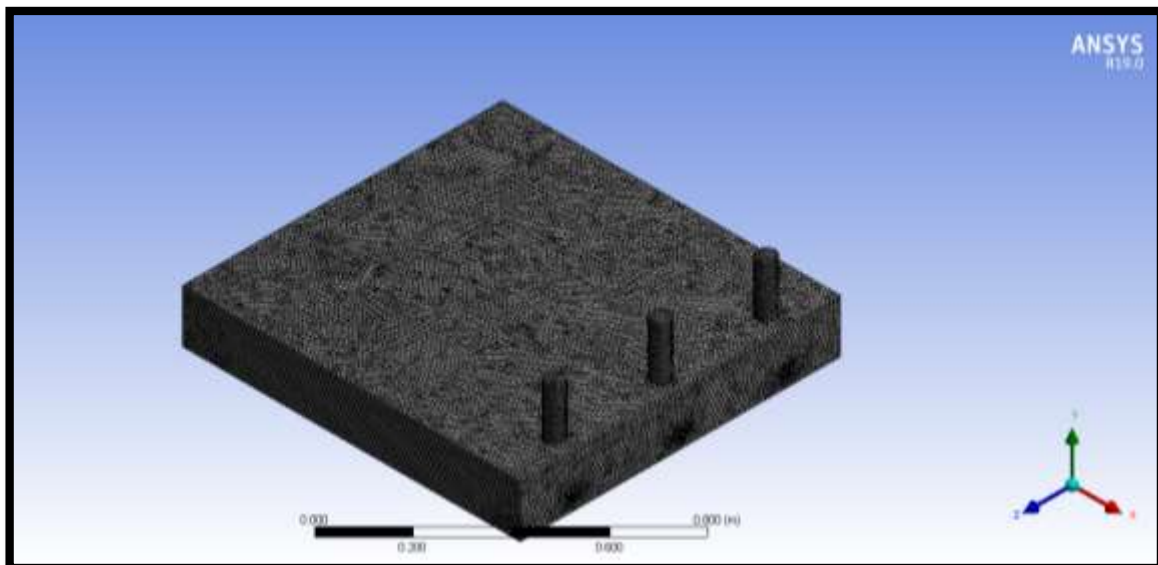
**Table 1.** Parameters of the study and its ratio.

Symbol	Description	Proposed ratio
<b>b</b>	Distance between intake pipes c/c	(2, 3, 4) $D$
<b>v</b>	Suction Velocity	(0.75, 1, 1.5, 2, 3) m/s
<b>S</b>	Submergence depth	(1, 1.25, 1.5, 1.75, 2) $D$



**Figure 2.** The geometry details.

**Fig. 3** shows the meshing of the geometry. Mesh generation refers to generate mesh for a numerical domain over a specified geometry to help the computer in solving the required equations. After the completion of the geometry, several computer-based software may be used to generate the mesh. In this study, ANSYS Fluent 19.0 was used to generate a high-quality tetrahedral mesh with a refinement grid in all the flow field.



**Figure 3.** The meshing of the geometry of pump sump and intakes.

## 2.2 The Numerical Simulation.

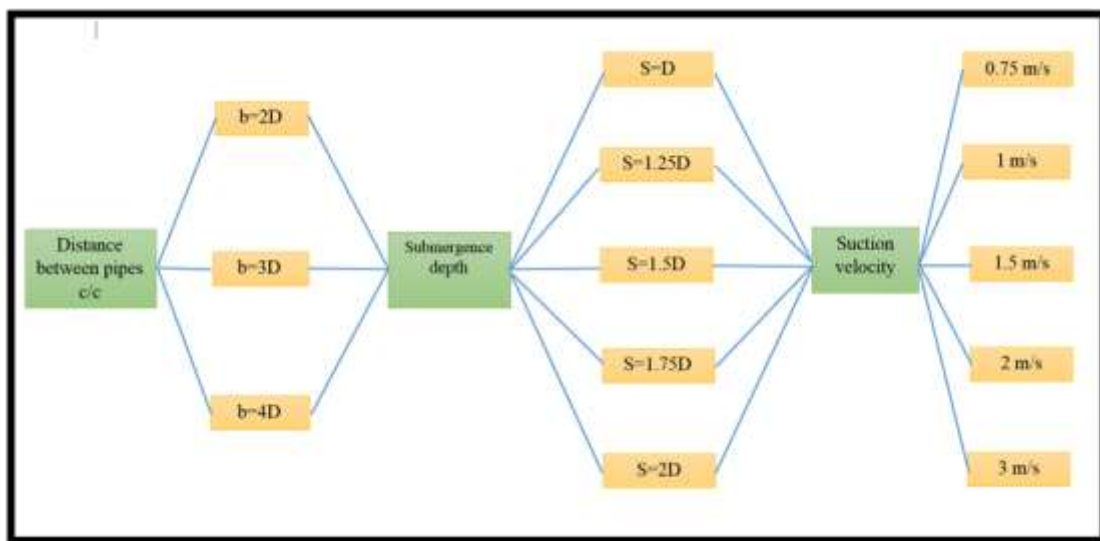
Simulation of the flows of fluids in hydraulic structures numerically considers a way to avoid the effect of scale that may exist in the laboratory tests of the hydraulic model. The solver of the flow which is used in this study is ANSYS Fluent 19.0 software. The flow through the pump sump and intakes is incompressible and turbulent. ANSYS Fluent solves the Navier-Stokes continuity equations numerically by finite volume method, which are based on principles of physics mass



conservation and Newton's Second Law within a moving fluid. Unsteady state analysis were done under different values of geometries dimensions and discharges for each model to cover the required cases of study. The flow properties were taken incompressible, 3D, and turbulent solving by the SST  $k-\omega$  model.

The Boundary Conditions, which was used in this analysis for the pump sump and intakes, are defined by the conditions of its inlet and outlet. The boundary condition of CFD analysis was the specified velocity inlet of the water in the inlet face of the tank for each case, suction velocity in the outlet of each suction pipe, and the top of the tank was free surface.

**Fig. 4** summarizes the plan of runs applied in this study. Seventy-five runs conducted to study the vortices formation and specify the critical submergence and drive formulae for it. The models were done for every distance between pipes ( $b$ ). Every distance between pipes modeled with five submergence depth ( $S$ ) as in **Table 1**, and every submergence depth runs with five different suction velocities ( $v$ ) as in **Table 1**.



**Figure 4.** Plan of runs for three pumps.

### 3. RESULTS AND DISCUSSION

The verification of the CFD model with the experimental results showed that the unsteady state of flow led to the best simulation for vortex. The observation showed that the vortex was not steady in the strength and position; it may be appeared, dissipated, and re-formed later and give a good coincidence with experimental visualization.

The results data of the numerical models collected to make comparisons between the cases of runs for discussion. The comparisons included clarification for the effect of different spaces between suction pipes on the formation of surface vortices and critical submergence. According to **ANSI/HI 9.8** standard, surface vortices entering the pump suction must be less than (Type 3). Therefore, critical submergence in this study refers to the free surface vortices (Type 4), which is likely to prevent its formation.

The results of the numerical simulation may be analyzed with many variables. In this study, it had been compared based on velocity profiles and the pattern of streamlines of flow. A cut plan is set on the surface of the water to study the behavior of the streamlines.

**Figs. 5 and 6** show the formation of air entering vortex based on streamlines and velocity magnitude. A large rotating in the mass of fluid was started in the free surface of the water, it



created a continuous gap in the surface of the water and reach the intake pipe with the pulling air, casing air entering the intake.

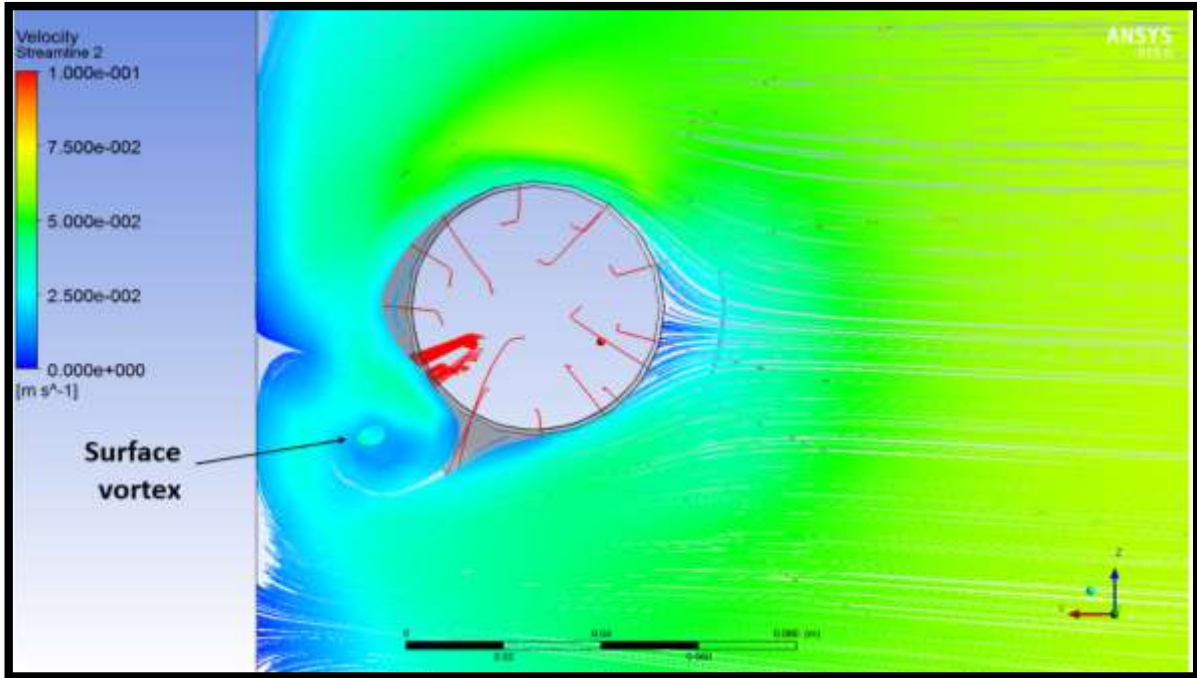


Figure 5. Surface vortex appeared in the free surface of water.

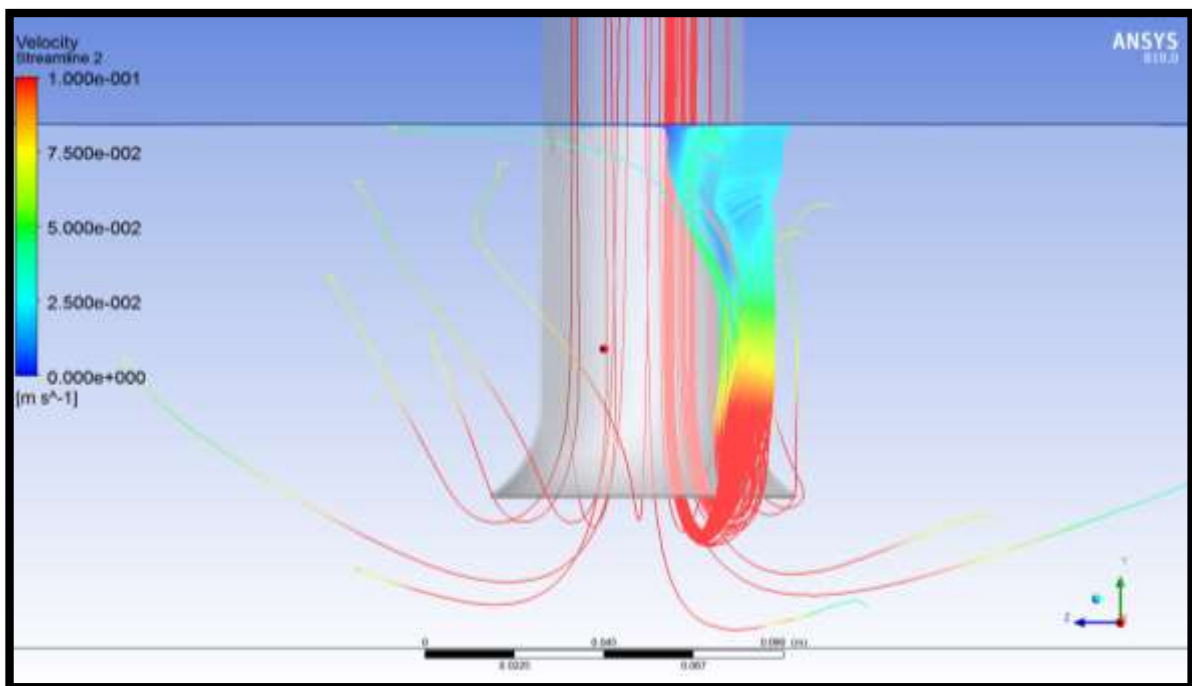


Figure 6. A surface vortex appeared in the sump side view.

To analyze the results and get the formulae of submergence depth of each space between pipes, the plot of the dimensionless ratio of submergence to bell mouth diameter versus intake Froude number at the bell mouth was presented for many model observations.



$$Fr = \frac{v}{\sqrt{gD}} \quad (1)$$

where:

$Fr$  : Froud Number, dimensionless.

$V$  : suction velocity (m/s).

$g$  : gravitational acceleration ( $m/s^2$ ).

$D$  : bell mouth diameter (m).

The plot is divided into two regions, a region (1) which lies below the critical line and includes the critical submergence, where intake vortices are unlikely and must be avoided. Region (2) is above the critical line of submergence with a good possibility of intake vortices, and a model study is recommended.

**Fig. 7** shows the critical submergence line for each space between intakes pipes that were studied. The result showed when (b) equal to  $2D$ , it was the danger cases between the three spaces. The pulling air vortex appeared when the submergence was ( $1.0D$ ,  $1.25D$ , and  $1.5D$ ) with any suction velocity that was specified in the study. The air entering vortex formed when the suction velocity equal or higher than  $1.5$  m/s when the submergence is  $1.75D$ , and it formed at submergence depth  $2D$  just when the suction velocity ( $v > 2$  m/s). The proposed equation gives relation for critical submergence ( $S_c$ ) as in **Eq. 2** with a coefficient of determination ( $0.996$ ).

$$S_c = D(0.4305Fr + 1.3258) \quad (2)$$

Besides, the analysis of results showed that the air entering vortex appeared clearly at submergence depth equal or less than  $1.25D$  when the space between suction pipes equal to  $3D$ . The pulling air vortex appeared just when the suction velocity ( $v \geq 1.5$  m/s) at submergence depth  $1.5D$ , and not formed with a velocity equal to or less than  $2$  m/s at submergence depth  $1.75D$ . **Eq. 3** determine the critical submergence for distance ( $3D$ ) with a coefficient of correction ( $1.0$ )

$$S_c = D(0.3607\ln(Fr) + 1.5938) \quad (3)$$

The best cases appeared when the space between the suction pipes is  $4D$ . The dangerous types of the vortex, which is pulling air, did not appear when the submergence depth greater than  $1.25D$ . Moreover, it is not formed with low velocities. For Froud number at bell mouth is equal or less than ( $1.028$ ), **Eq. 4** can be used to compute the critical submergence with the coefficient of correction ( $1.0$ ). The results showed that the ratio of dimensions of ( $S/D$ ) still constant at  $1.25$  when the Froud number is more than ( $1.028$ ).

$$S_c = D(0.9727Fr + 0.25) \quad (4)$$



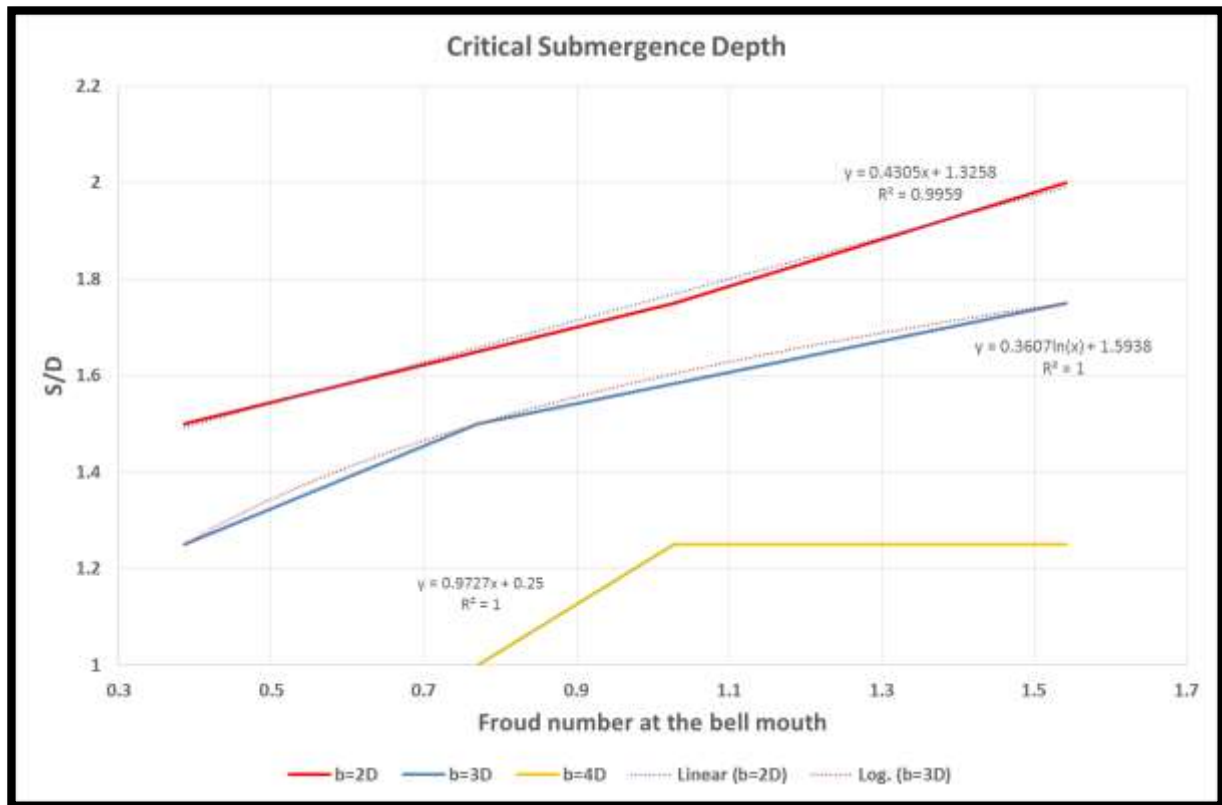


Figure 7. Critical submergence for each space between pipes.

#### 4. CONCLUSIONS

In this study, the effect of space between intake pipes with some limitations of geometry dimensions and suction velocities was investigated numerically by CFD to get critical submergence at three intakes. Most surface vortices of (Type 4) are pulling air and lead to the occurrence of cavitation phenomena, which can cause noise and vibration. This may indicate that the surface vortices are more dangerous on the pumping system. The results showed that:

1. There is a clear relationship for the distances between the intakes on the formation of the vortex (Type 4), formed as an air gap sucks air to the pump intake continuously. The distance (4D) get the best operation cases when the distance (2D) was the worst.
2. When the space between intakes was (2D), the vortex (Type 4) appeared obviously at the submergence ( $S \leq 2D$ ) and the suction velocity greater than (2 m/s).
3. The vortex (Type 4) did not form at submergence greater than (1.25D) when the space between pipes equal to (4D), and not appear for velocity less than (2 m/s) at submergence (1.25D)
4. Results of the numerical model and the formulae introduced in (Fig. 7) for each space between pipes (b) can be useful for designing intakes. It helps in avoiding the problems that may happen as a result of air entering from intakes and reach pumps.



## REFERENCES

- Ahmadi, H., and Razavi, E., 2018. Determination of Submergence Depth to Avoid Vortices at Horizontal Intake Applying Flow-3D Software. *Journal of Civil Engineering Research*, 8 (3), pp., 62-69.
- Al-Nakeeb, A., Al-Samawi, A., A., and Al-Saffar, H., A., 2018. Upgrading of Alum Preparation and Dosing Unit for Sharq Dijla Water Treatment Plant by Using Programmable Logic Controller System. *Journal of Engineering*, 24 (2), pp., 131-141.
- Athab, A., H., Al-Khateeb, H., M., M., and AL-Thamiry, H., A., 2017. Investigating the Setting of the Entrance of Multiple Vertical Suction Pipes in Storm Water Pump Station. *International Advanced Research Journal in Science, Engineering and Technology*, 4 (8), pp., 69-78.
- Gulliver, J.S., Rindels, A.J. (1987). Weak vortices at vertical intakes. *J. Hydraul. Eng. ASCE*, 113(9), pp., 1101–1116.
- Guo, M., Chen, Z., and Choi, Y., 2019. Effect of suction pipe inlet condition on the occurrence of vortex in pump sump. *IOP Conf. Series: Earth and Environmental Science* 240 (2019) 062023.
- Hydraulic Institute Standards, 1998. *American National Standard for Pump Intake Design*, ANSI/HI 9.8.
- Kabiri-Samani, A., R., and Borghei, S., M., 2013. Effects of anti-vortex plates on air entrainment by free vortex. *Scientia Iranica*, 20 (2), pp., 251-25.
- Khanarmuei, M., Rahimzadeh, H., and Sarkardeh, H., 2018. Effect of dual intake direction on critical submergence and vortex strength. *Journal of Hydraulic Research*, [Online] Available at: (<http://www.tandfonline.com/loi/tjhr20>).
- Kleynhans, S., 2012. Physical hydraulic model investigation of critical submergence for raised pump intakes. Master thesis. Stellenbosch University, Department of Civil engineering.
- Moreno, C., J., G., 2014, Determining Critical Submergence in Tanks by Means of Reynolds & Weber Numbers. *World Journal of Engineering and Technology*, [Online] Available at: <http://dx.doi.org/10.4236/wjet.2014.23024> .
- Salal, A., M., and Khudair, B., H., 2019. Influent Flow Rate Effect On Sewage Pump Station Performance Based On Organic And Sediment Loading. *Journal of Engineering*, 25 (9), pp., 1-11.
- Szeliga, N., Elgott, L., H., Bezecny, D., Richter, S., Hoffmann, M., and Schluter, M., 2019. Large-Scale Experiments on the Formation of Surface Vortices with and without Vortex Suppression. *WILEY-VCH Verlag GmbH & Co. KGaA, Weinheim*, 91 (12), pp., 1-11.
- Werth, D., and Frizzell, C., 2009. Minimum pump submergence to prevent surface vortex formation. *Journal of Hydraulic Research*, 47 (1), pp., 142-144.
- Zhao, Y., 2014. Study of Flow Field in Rectangular Sump Models and Performance Analysis of a Mixed Flow Pump. Master Thesis, Department of Mechanical Engineering, Korea Maritime and Ocean University, Korea.