

Free Surface Vortices at Intakes: Influence of Different Intake Geometries on Critical Submergence and Air Entrainment

Conference Paper**Author(s):**

Schneider, Lukas M.; Farshi, Davood; De Cesare, Giovanni; Piroozmand, Pasha

Publication date:

2024

Permanent link:

<https://doi.org/https://doi.org/10.3929/ethz-b-000675967>

Rights / license:

[Creative Commons Attribution 4.0 International](#)

Free Surface Vortices at Intakes: Influence of Different Intake Geometries on Critical Submergence and Air Entrainment

L.M. Schneider¹, D. Farshi², G. De Cesare³, P. Piroozmand²

¹AFRY Switzerland Ltd., Zurich, Switzerland

²OST, Institute for Civil and Environmental Engineering, Rapperswil, Switzerland

³LCH, EPFL, Lausanne, Switzerland

E-mail: lukas.schneider@afry.com

Abstract: The formation of free surface vortices at intakes results in air entrainment into the headrace system of hydropower plants. This reduces efficiency, may damage turbines and pumps, and increases the likelihood of transient phenomena in pressurized waterways. To prevent these losses and damages, operators of hydropower plants maintain a minimum submergence level above the intake. As a result, a significant part of storage volume is not usable leading to lower energy storage capacity and income. In this study, a two-phase 3D CFD model was implemented in OpenFOAM to examine the influence of five different intake geometries on air entrainment and critical submergence. For this purpose, eleven different model configurations were used. Among different RANS turbulence models, the $k-\omega$ SST model was chosen. For validation of the CFD model, the data of a physical model were compared with the simulation results. It can be shown that, in the case of low circulation above the intake, unsteady RANS models cannot properly capture the dynamics of free surface vortices. The RANS approach tends to overestimate the turbulent kinetic energy and the turbulent viscosity of the weak circulation region above the intake induced by the geometry where the kinetic energy is indeed low. This results in excessive kinetic energy losses and the already weak circulation in the model is not sufficient to develop either surface dimple formation or full air core vortices. For two model configurations, the simulations show reasonable results for the estimation of the air entrainment. Furthermore, the results confirm that the distance of the intake opening from the front wall and the inclination of the front wall have a significant influence on air entrainment and critical submergence.

Keywords: free surface vortex, intake, air entrainment, CFD, two-phase 3D model, two-equation turbulence model

List of Symbols and Abbreviations:

CC	Curvature Correction
CFD	Computational Fluid Dynamics
DNS	Direct Numerical Simulation
F_{co}	Combined Froude number ($F_{co} = U/\sqrt{gh}$)
F_D	Inlet Froude number ($F_D = U/\sqrt{gD}$)
FVM	Finite Volume Method
$k-\varepsilon$	Two-equations turbulence model based on turbulent kinetic energy and the rate of dissipation of turbulent kinetic energy
$k-\omega$	Two equations turbulence model based on turbulent kinetic energy and specific rate of dissipation
LES	Large Eddy Simulation
RANS	Reynolds-averaged Navier-Stokes
RSM	Reynolds-Stress Model
SST	Shear Stress Transport
VOF	Volume of Fluid

1. Introduction

Air entrainment reduces the efficiency of turbines and pumps and can lead to their damage. The damages range from vibration, noise, cavitation, rotation stall, and material fatigue (Azman, et al., 2020). To prevent air entrainment, the phenomenon of vortex formation must be understood, and its most influencing parameters must be recognized. There are still gaps in the current state of knowledge in this regard. In addition to air, free surface intake vortices can also transport ice crystals, which adhere to the intake screen. Within a short time, significant ice layers are established,

which can lead to the complete blockage of the intake (Martinerie, Müller, De Cesare, & Boillat, 2008). Furthermore, the progressive sedimentation of storage volume at existing hydropower plants has led to the need for lifting the intake in order to have sufficient distance to the sediment deposits. Due to this situation, the question of the minimum water level in the reservoir and the critical submergence also arises for existing reservoirs. The purpose of this study was to find out whether the phenomenon of free surface vortices can be reproduced by RANS approaches in OpenFOAM.

2. State of the Art review

2.1. Vortex Formation

Knauss (1987) describes the eccentricity or asymmetry of the inflow to the intake as the most important cause for the formation of intake vortices. Under certain conditions, this asymmetry can occur in the inflow even with symmetrically designed intakes. For example, if there are irregularities in the roughness of the border or if the inflow into the reservoir is eccentric. Figure 1 shows examples of the formation of intake vortices due to asymmetrical inflow conditions. An immersion wall located in front of the intake, a bottom sill, a bridge pier, or the influence of wind and waves as well as backflows can also lead to asymmetrical inflow conditions. This asymmetry induces a rotating flow over the intake. Hecker describes in Knauss (1987) the formation of an intake vortex. Accordingly, the inflow conditions result in a maximum tangential velocity in the rotating flow above the intake. The velocity height from the tangential velocity leads to a loss of hydrostatic pressure, causing the water surface to sink in the center of the rotation. This lowering of the water level can range from a small subsidence ("dimple", type 2) to an intake vortex with a fully developed air core reaching the intake cross section (type 6, see Figure 2).

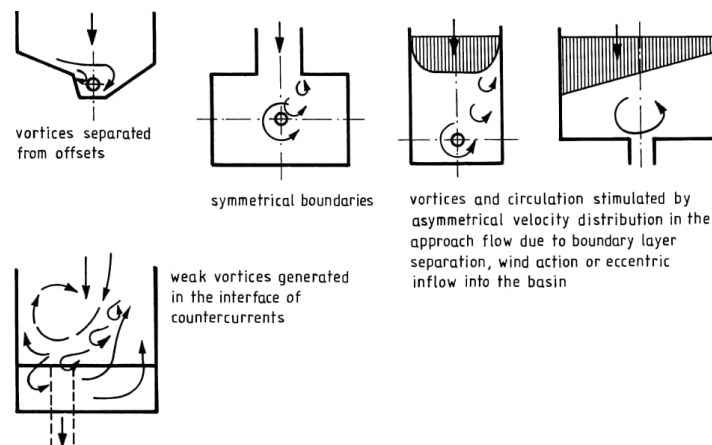


Figure 1. Selected examples of the formation of intake vortices due to asymmetrical approach flow conditions (Knauss 1987).

Padmanabhan and Hecker (1984) used the classification of intake vortices into six different types, which is still in use today. A distinction is made between a slight rotation on the surface (type 1) and a fully formed intake vortex with an air core up to the intake cross-section (type 6) (see Figure 2).

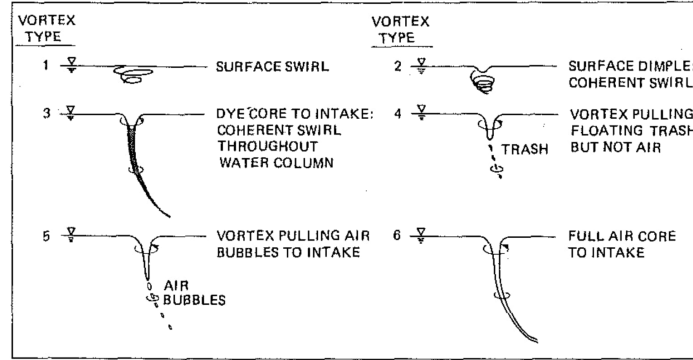


Figure 2. Six types of intake vortices (Padmanabhan and Hecker 1984).

2.2. Critical submergence and air entrainment

Based on a physical model with a horizontal intake (see Figure 3), Möller (2013) derived a function describing the air entrainment $\bar{Q}_a$ with respect to the discharge Q_w and the relative submergence h/D . It describes the mean air entrainment $\bar{Q}_a$ over phases with and without a fully developed free surface intake vortex with a coefficient of determination of $R^2 = 0.97$.

$$\bar{Q}_a = 0.005 e^{64 \bar{Q}_w / \sqrt{h/D}} \quad (1)$$

The dimensionless air entrainment rate β is the ratio of the air entrainment Q_A and the discharge Q_w in the intake ($\beta = Q_A/Q_w$). For the mean air entrainment rate $\bar{\beta}$, Möller (2013) describes the following relationship with a coefficient of determination of $R^2 = 0.90$. Here the combined Froude number is considered instead of the inlet Froude number. The relationship is validated for $0.3 \leq F_{co} \leq 0.8$.

$$\bar{\beta} = 1.04 \cdot 10^{-8} e^{16 F_{co}} \quad (2)$$

Furthermore, Möller (2013) showed that the circulation of the intake vortex Γ plays a minor role in air entrainment.

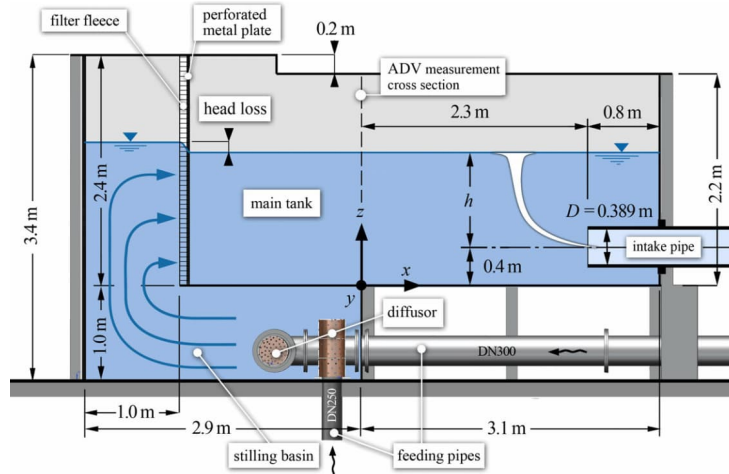


Figure 3. Longitudinal section of the experimental tank (width = 1.6 m), with feeding pipes, stilling basin, and intake pipe (Möller, 2013).

The critical submergence height h is measured from the center of the intake cross-section to the water surface.

2.3. Influence of the inclination of the front wall

Sarkardeh, Zarrati and Roshan (2010) investigated the influence of the inclination of the front wall over a horizontal intake on vortices in a physical model. They were able to show that the type of intake vortex and its strength decreases with increasing steepness of the front wall.

2.4. Numerical Modelling

Domfeh et al. (2020) reproduced the physical model of Sarkardeh, Zarrati, Jabbari and Tavakkol (2013) in a numerical model where the model geometry was perfectly symmetrical. Domfeh et al. (2020) used the RANS approach with the stationary LTSinterFoam solver in OpenFOAM and investigated different turbulence models. To capture the interface between water and air, the VOF method was applied. According to their findings, the $k-\omega$ SST turbulence model achieves the best agreement with the results of the physical model among the investigated turbulence models. They identified a research gap in the field of transient simulations of the occurrence of intake vortices with interFoam in OpenFOAM. This gap is partially tackled with the present publication.

Several authors investigated intake vortices with numerical two-phase models using the interface capturing method VOF for asymmetric geometries with vertically aligned floor openings. These were, for example, Chan, Guo and Lee (2021) with the software ANSYS FLUENT and the $k-\varepsilon$ CC turbulence model and Mulligan (2015) with ANSYS CFX and $k-\varepsilon$, $k-\omega$ SST, RSM and other turbulence models. In these cases, the rotation above the intake was given due to the asymmetry of the model domain. Chan, Guo and Lee (2021) were able to find a very good agreement of the tangential flow velocity and the circulation from the numerical model and the physical model (coefficient of determination $R^2 = 0.80 - 0.99$). According to Mulligan (2015), a transient solver leads to better results at the free water surface and to more accurate tangential velocity profiles compared to the results of a stationary solver. However, the flow depth in the vortex chamber is underestimated. For an inlet Froude number of approximately $F_D = 0.4$ by around 2.5% and for $F_D = 0.65$ by 11%. Mulligan (2015) showed that using the RANS approach and RSM the highest accuracy is achieved with 2-3% deviation, and $k-\omega$ SST CC still gives the second-best results with 12% deviation in water level and 22% in tangential velocity. More accurate results are only attributed to the turbulence model LES and to DNS, but with much higher computational effort. In addition, Mulligan (2015) stated that the mesh size at the orifice should be no greater than 1.5% of the outlet diameter to obtain mesh-independent numerical results in the area of the expected vortex.

Sarkardeh (2017) reproduced the physical model of Möller (2013) in a numerical model with the software Flow 3D and an LES turbulence model. The cell size in the intake area was set to 2 mm. Depending on the configuration, the simulations show fully formed intake vortices of type 6 (see Figure 4).

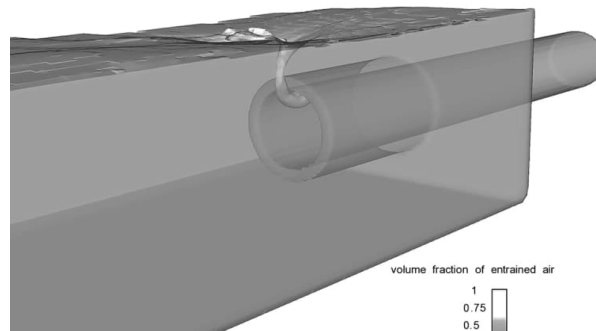


Figure 4. Simulation of a free surface vortex at $h/D = 1.0$ and $F_D \sim 0.6$ with the software Flow 3D and an LES turbulence model (view in the direction of flow, Sarkardeh 2017)

Furthermore, Sarkardeh (2017) compared the mean air entrainment rate from his investigations with the results of Möller (2013). He found that the simulation tended to result in air entrainment rates that were too high for low combined Froude numbers F_{co} and too low for medium ones. In the case of strong intake vortices (type 6), the mean deviation from the air entrainment rate of Möller (2013) is approx. 29%. Sarkardeh (2017) determined lower values for critical submergence than those determined in previous publications using data from prototypes and physical models. Accordingly, in a comparison with previous design approaches with the toleration of an air entrainment rate of $\beta = 10^{-5}$, the critical submergence could be reduced by at least 12%. In addition, he describes that the results for the circulation are also consistent with physical model results and using an LES model is appropriate with respect to

circulation investigations. Despite the above findings, showing a reasonable applicability of the described simulation approach, the authors of the present publication point out that the geometry of the vortex, in explicit the diameter of the core, is overestimated and deviates from reality (see Figure 4).

2.5. Governing Equations

The motion of a fluid is basically described by a system of five partial differential equations: the continuity equation, the momentum equations in x, y, and z direction, and the energy equation. The system can be reduced to the continuity equation and the momentum equations, neglecting heat transport and with the presumption of incompressible fluids, which is valid for gases at low flow velocities (Versteeg & Malalasekera 2007).

With the above constraints, the underlying equations for continuity (3) and momentum exchange in all three directions of space (4) can be written as follows:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (3)$$

$$\frac{\partial \rho u_i}{\partial t} + u_j \frac{\partial \rho u_i}{\partial x_j} = -\frac{\partial p^*}{\partial x_i} - g_j x_j \frac{\partial \rho}{\partial x_i} + \frac{\partial}{\partial x_j} (2\mu S_{ij}) + \sigma_T \kappa \frac{\partial \alpha}{\partial x_i} \quad (4)$$

$$S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (5)$$

Where u_i stands for the mean velocity components in the x, y, and z direction, x_i for the cartesian coordinates, ρ for the density of water or air, p^* for the pressure minus the hydrostatic component ($\rho g_j x_j$), g_j for the gravitational acceleration, $\mu = \rho \nu$ for dynamic molecular viscosity, and S_{ij} for the mean strain rate tensor. The last term of equation (4) contains the surface tension σ_T and the curvature of the surface κ .

3. Experimental Setup and Simulation

3.1. Physical Model as Reference

Figure 3 illustrates the setup of the physical model of Möller (2013). He investigated 34 different model configurations. The cases ranged from $h/D = 1.25 - 2.5$ and from inlet Froude numbers $F_D = 0.4 - 1.2$, corresponding to discharges of 92.8 l/s and 278.6 l/s, respectively. Möller (2013) was able to produce fully developed free surface vortices (type 6, see Figure 2).

3.2. Numerical Model

The numerical two-phase (water-air) simulations of the present study contain eleven different model configurations. The values for the relative submergence h/D and inlet Froude number F_D of the cases investigated were placed within the ranges of the model setups of Möller (2013) in order to gain comparable results.

3.2.1. Base Model

Figure 5 illustrates the setup of the base model. The model inlet is at the bottom left and slightly shifted from the symmetry axes, to introduce an asymmetry to the flow. The model inlet area was chosen to be impounded by water, which increases the numerical stability of the simulations. In this specific case (base model) the tank reaches up to a height of 1.0 m, where a lateral widening is visible. From this level upwards, there is atmosphere at the sides and the top area. In the transition area from wall to atmosphere lies the interface between water and air. The intake pipe is located 0.40 m above the ground. The plan view of the tank and the position of the intake as well as the inner and outer diameters of the intake pipe were adopted 1:1 from the physical model of Möller (2013).

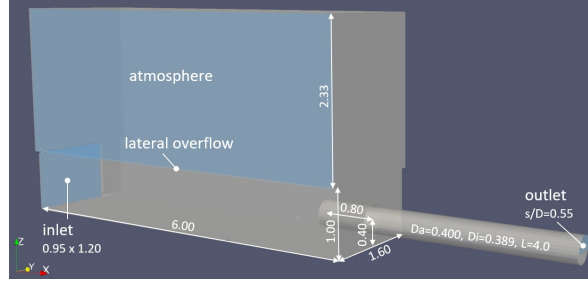


Figure 5. Isometric view of the base model to illustrate the model set-up with an asymmetrically arranged model inlet. Measures in [m], $h/D = 1.5$.

Möller (2013) regulated the outflow of the model with a valve in the outlet pipe. Analogous, a reduction of the cross-section was introduced at the outlet cross section to regulate the model outflow.

At both sides of the tank the water was able to overtop the lateral edge, where the tank walls meet the atmosphere. The discharge at the model inlet was chosen slightly higher than at the expected model outlet discharge. With this setup a constant water level in the tank is maintained and hence a constant discharge at the intake is guaranteed.

3.2.2. Analyzed Intake Geometries

Since the arrangements of intakes for hydro power plants are manifold, the current study aimed to get insights of the influence of common intake geometries on the air entrainment. Therefore, in addition to the base model configuration, the following four simplified intake geometries were investigated (see Figure 6).

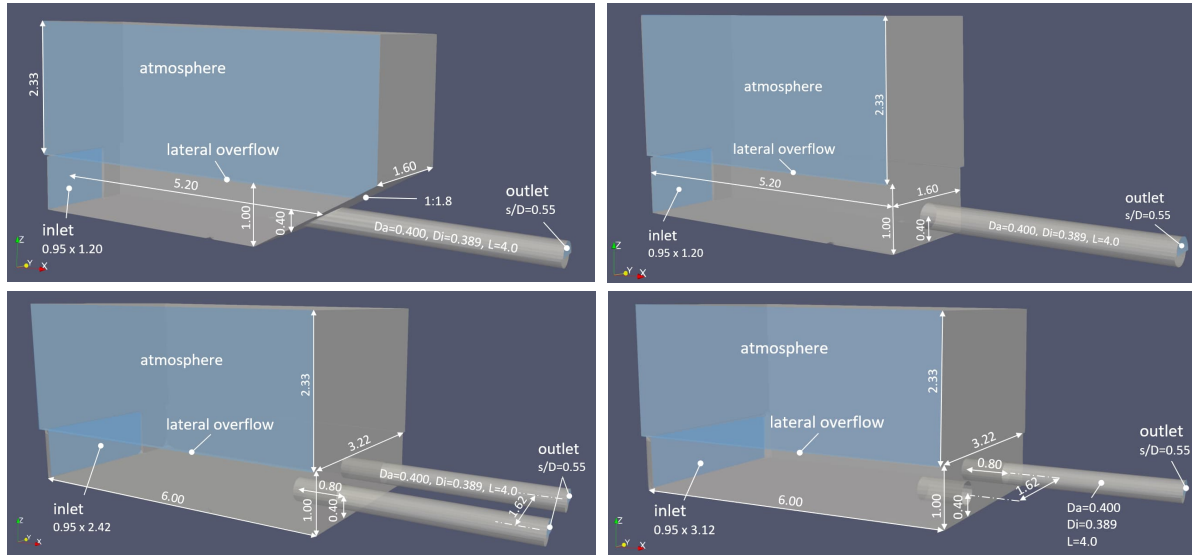


Figure 6. Isometric views of the investigated intake geometries. From top left to bottom right: inclined front wall (1:1.8) and flush intake (A), vertical front wall and flush intake (B), two open intakes and one open intake (C). Measures in [m], $h/D = 1.5$.

3.2.3. Software, Solver and Turbulence Model

OpenFOAM is an open source CFD package for various applications. The transient solver InterFoam was applied. Due to its good simulation stability and general accuracy, the turbulence model $k-\omega$ SST was chosen.

3.2.4. Boundary and Initial Conditions

As boundary conditions, the following setup was used (see Table 1).

Table 1. Boundary Conditions

Field Property		Inlet	Outlet	Wall	Atmosphere
Flow Velocity	U	flowRate-InletVelocity	zeroGradient	noSlip	pressureInlet-OutletVelocity
Reduced Pressure	p_{rgh}	fixedFlux-Pressure	fixedValue	fixedFlux-Pressure	fixedValue
Phase Fraction	α_{water}	fixedValue	zeroGradient	zeroGradient	inletOutlet
Turbulent Kinetic Energy	k	fixedValue	inletOutlet	kqRWall-Function	inletOutlet
Specific Dissipation Rate of k	ω	fixedValue	inletOutlet	omegaWall-Function	inletOutlet
Turbulent Viscosity	ν_t	calculated	calculated	nutkRough-WallFunction	calculated

As initial conditions the velocity U at the inlet was defined by the discharge, which was chosen between 0.24 and 0.30 m³/s, pressure p_{rgh} was set to 0 Pa at the model outlet and the atmosphere.

3.2.5. Mesh

The meshing tool snappyHexMesh of OpenFOAM generates an unstructured mesh with mostly hexahedron-shaped cells. The geometry, i.e. the ratio of the side length of a cell, is specified in advance by the blockMesh tool. The cell size in the intake pipe was iteratively evaluated to place y^+ well above 30 and below 130 to respect the wall function. To keep the total number of cells at a manageable level, the cell size was stepwise refined towards the intake and the area of the expected vortex. A sensitivity analysis was conducted to find an optimum cell size, which still allowed to reach accurate results for air entrainment with reasonable computational efforts. With this procedure, a minimum cell size of 10 mm was found reasonable for the purpose of the present study.

4. Results and Discussion

4.1. Comparison of physical and numerical Model Results

4.1.1. Air Entrainment

A comparison of the air entrainment $\bar{Q}_a$ and the air entrainment rate $\bar{\beta}$ with the findings of Möller (2013) is done for the base model configurations. Due to the lack of better data, the results of the model configurations with modified intake geometry are also compared with Eq. (1) and (2) according to Möller (2013) to assess the plausibility of the magnitude.

The following Figure 7 and Figure 8 summarize the results of all eleven simulations. The two most consistent values for the air entrainment rate compared to Möller (2013) are found for the intake geometries with a flush intake and an inclined front wall (A), and a flush intake with vertical front wall (B). The air entrainment $\bar{Q}_a$ of the model configuration A (with $h/D = 1.61$, $F_D = 0.71$) is about 29% higher than the value according to Eq. (1) and for B (with $h/D = 1.63$, $F_D = 0.74$), the air entrainment is about 24% below Eq. (1).

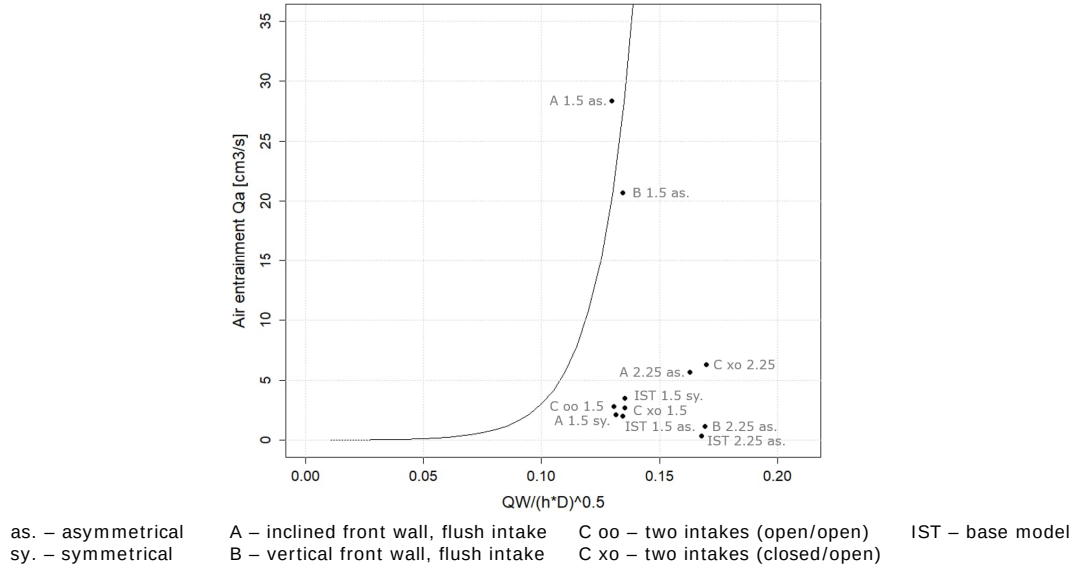


Figure 7. Comparison of the results from the present study (points) to the relationship between air entrainment $\bar{Q}_a$, flow rate Q_W and submergence h according to Eq. (1) of Möller (2013) (line).

The air entrainment rate $\bar{\beta}$ for the model configuration B (with $h/D = 1.63$, $F_D = 0.74$) is very close to the line according to Eq. (2) and for A (with $h/D = 1.61$, $F_D = 0.71$) reaches the limit of the upper range of the 95% prediction interval.

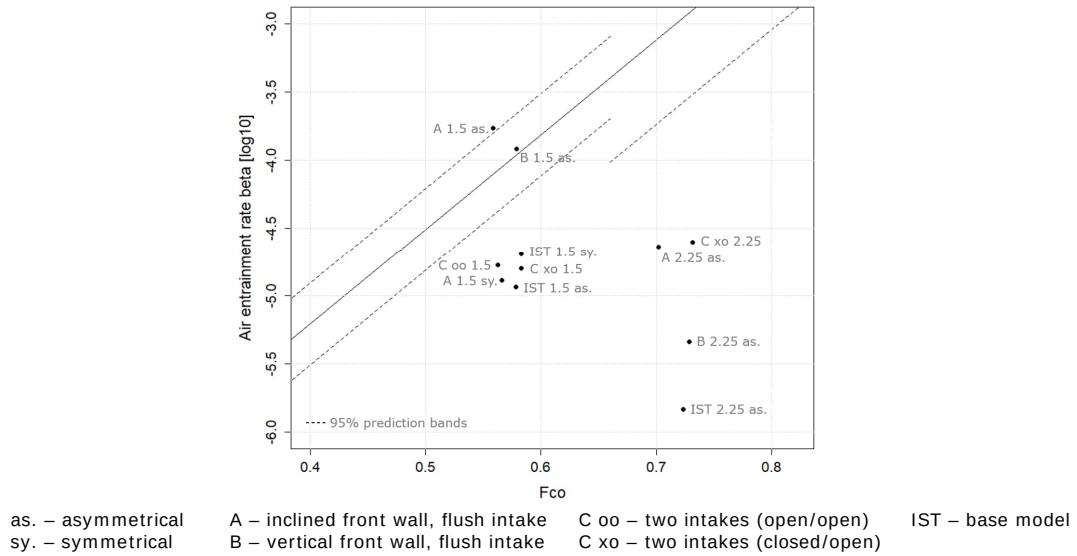


Figure 8. Comparison of the results from the present study (points) to the prediction bands (95%) for the air entrainment rate $\bar{\beta}$ according to Eq. (2) of Möller (2013) (line). The vertical axis is logarithmic with base 10 and shows the exponents of the power of ten.

The results for the air entrainment and the air entrainment rate of the two simulations with a flush intake are judged to be plausible, based on the comparison of the order of magnitude with the equations (1) and (2) of Möller (2013). Besides for the base model, an actual 1:1 comparison of the results from the numerical simulations and the physical model is not allowed for the modified model geometries (see Figure 6).

4.1.2. Circulation

For all eleven simulations with different model configurations no free surface vortex was resolved by the model. In the following, the findings for circulation above the intake are discussed.

For the base model with an asymmetrically arranged inlet opening (see Figure 5), there is a good agreement between the position of the center of rotation in the intake area and the results of the physical model by Möller (2013) (see Figure 9). In both cases, the center of rotation is located within an area of approx. 0.4×0.4 m in the upper left corner (in plan view). This indicates that the introduced asymmetry was chosen appropriately and in a way that excessive turbulence in the tank is prohibited. The intake submergence is identical for the compared numerical and the physical model. The discharge in the numerical model is 252 l/s, which is about 20% higher than in the physical model. Even though the discharge in the numerical model is higher, no free surface intake vortex was detected.

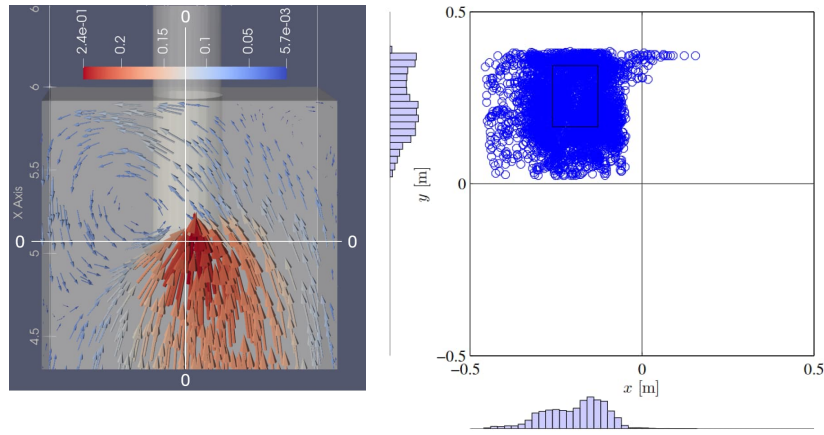


Figure 9. Center of circulation in the numerical model with visualized velocity vectors U_{xy} [m/s] (left). Variation of the center of the inlet vortex over the duration of the experiment in the physical model of Möller (2013) (plan view at the intake opening, 0/0 denotes the position of the intake, flow direction is from bottom to top). Measures in [m].

Figure 10 shows a comparison of the circulation above the intake of the base model with an asymmetrically aligned model inlet and the models with an inclined front wall (1:1.8) and flush intake (A) and with a vertical front wall and flush intake (B). In contrast to the base model and case A, case B exhibits no circulation above the intake. It illustrates the influence of the horizontal distance of the front wall to the inlet cross-section of the intake. The closer the front wall, the more likely is the suppression of the circulation. For a flush intake with a vertical front wall (B), the occurrence of a free surface vortex is prohibited to some extent. Hence, the critical submergence can be lower than for a flush intake with an inclined front wall and a protruding intake as in the base model. In the base model, the circulation above the intake is almost not disturbed, so the critical submergence must be the largest among the three compared intake geometries. This finding is in line with the results of the study on the inclination of the front wall in a physical model investigation by Sarkardeh et al. (2010).

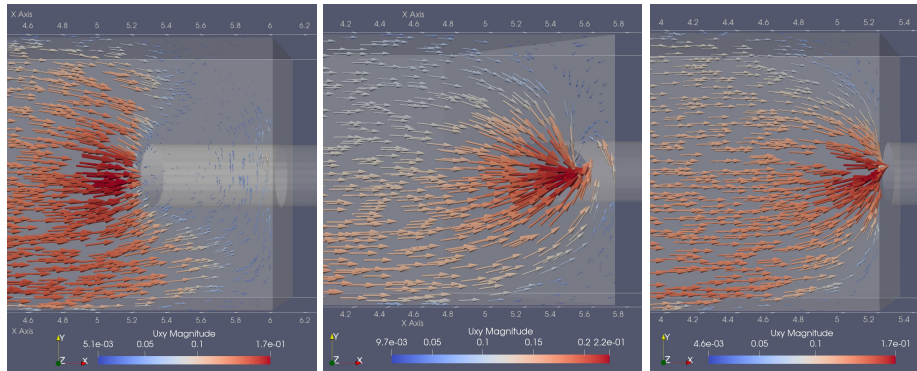


Figure 10. Plan view on the intake area of the base model (asymmetrical) (left) and model configuration A (middle) and B (right) with visualized velocity vectors U_{xy} [m/s] at a height of approx. 30 cm above the intake axis ($h/D = 1.62, F_D = 0.74$; $h/D = 1.61, F_D = 0.71$; $h/D = 1.63, F_D = 0.74$). Measures in [m].

5. Conclusion

Based on this study, the following conclusions are made:

- The investigations show that the distance of the intake opening to the front wall and the inclination of the front wall have a significant influence on air entrainment and critical submergence. For example, the air entrainment was around 27% higher for the model configuration with an inclined front wall and a flush intake than for a vertical front wall with a flush intake. As a result, the critical submergence is higher with an inclined front wall.
- The comparison of the simulation results for the air entrainment of two adjacent intakes, with an axis clearance of $4.2D$, where both or only one was in operation, shows no significant influence.
- For two (A 1.5 as. and B 1.5 as.) out of eleven simulations, plausible air entrainment and air entrainment rates were found. In the other nine simulations, the air entrainment was significantly too low. It is assumed that the use of the RANS approach with the $k-\omega$ SST turbulence model leads to inaccurate results in case of weak circulation above the intake due to its diffusive nature compared to LES.
- This study confirms that using the RANS approach for a perfectly symmetrical model geometry with horizontal intake does not allow the formation of circulation above the intake and therefore the formation of free surface vortices is prohibited. As a consequence, the model geometry was slightly asymmetrically altered.
- The comparison of the air entrainment in models with and without rotating flow above the intake did not reveal a clear correlation. Therefore, it is assumed that for small air entrainment quantities, circulation above the intake is not a prerequisite to determine the air entrainment in a numerical model.
- Even though, no free surface vortex formation was detected in the simulations, two results align reasonably well with physical data. It is assumed that the source of the detected air lies in the interface of air and water, where small quantities of air are transferred into the water phase and transported towards the intake. This assumption is supported by the impression gained while tracking the path of cells containing a substantial portion of air. These cells emerge from the water surface and follow the velocity vectors down to the intake (see Figure 11). We assume the lack of surface vortex formation is due to the diffusive nature of the RANS model where the small eddies are modeled with eddy viscosity.

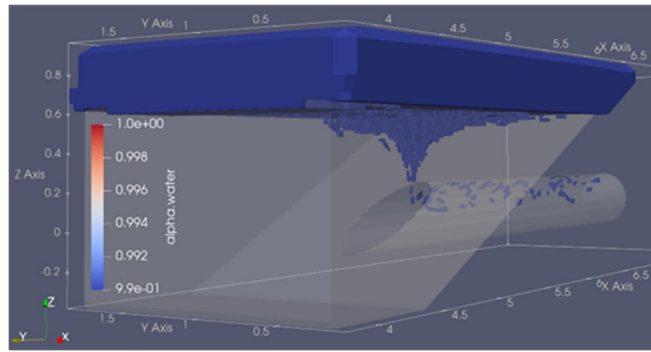


Figure 11: Visualization of cells with an air content larger than 1% (blue) in the model configuration A with the inclined front wall and flush intake ($h/D = 1.61$, $F_D = 0.71$). View in the direction of flow. The water phase is hidden. Measures in [m].

- For all eleven model configurations, no fully developed free surface vortex with an air core down to the intake structure was detected. On the one hand, this is attributed to the weak circulation above the intake due to the model geometry, and on the other hand, the RANS approach is known to overestimate the turbulent kinetic energy. This may result in too high energy losses and the already weak circulation in the investigated models is no longer sufficient to produce fully developed free surface vortices. The use of the RANS approach with a $k-\omega$ SST turbulence model seems to reach its limits with these prerequisites. It is therefore assumed that there must be a minimum circulation above the intake, to successfully simulate free surface vortices using the RANS approach.

Since the RANS approach is still more cost-efficient in numerical modeling than LES or DNS, it is recommended that further studies explore the necessary limit of circulation above the intake structure to be able to reproduce reasonable air entrainment rates with the RANS approach.

6. ACKNOWLEDGMENTS

This work was conducted in collaboration with the Institute for Construction and Environment (IBU) of the University of Applied Sciences of Eastern Switzerland OST and the Laboratory of Hydraulic Constructions (LCH) of the Ecole Polytechnique Fédérale de Lausanne EPFL. This study was not supported by any funding sources.

7. REFERENCES

- Azman, A., Zahari, N., Mansor, M., Ng, F., Abas, A., Nurhikmah, F., & Aziz, N. A. (2020). The Impact of Vortex Formation due to the Operational Dam Condition: A Review. *IOP Conference Series: Materials Science and Engineering*, 1-6.
- Chan, S., Guo, J., & Lee, J. H. (2021). Physical and numerical modeling of swirling flow in a scroll vortex intake. *Journal of Hydro-environment Research*, 64-76.
- Domfeh, M. K., Gyamfi, S., Amo-Boateng, M., Andoh, R., Ofori, E. A., & Tabor, G. (2020). Numerical Simulation of an Air-Core Vortex and Its Suppression at an Intake Using OpenFOAM. *Fluids*, 1-12.
- Knauss, J. Ed. (1987). Swirling Flow Problems at Intakes. IHAR International Association for Hydraulic Research. Taylor & Francis, New York.
- Martinerie, R., Müller, M., De Cesare, G., & Boillat, J.-L. (2008). Vortices at intake works of pump-storage schemes. *Proceedings of the 6th International Symposium on Ultrasonic Doppler Method for Fluid Mechanics and Fluid Engineering*, Prague, 119-122.
- Möller, G. (2013). Vortex-Induced Air Entrainment Rate at Intakes, Mitteilungen 220. Versuchsanstalt für Wasserbau, Hydrologie und Glaziologie (VAW), ETH Zürich.
- Mulligan, S. (2015). Experimental and Numerical Analysis of Three-Dimensional Free-Surface Turbulent Vortex Flows with Strong Circulation. Institute of Technology, Sligo ITS.
- Padmanabhan, M., & Hecker, G. E. (1984). Scale Effects in Pump Sump Models. *Journal of Hydraulic Engineering*, 1540-1556.
- Sarkardeh, H. (2017). Numerical calculation of air entrainment rates due to intake vortices. *Meccanica*, 3629-3643.
- Sarkardeh, H., Zarrati, A. R., & Roshan, R. (2010). Effect of intake head wall and trash rack on vortices. *Journal of Hydraulic Research*, 108-112.
- Sarkardeh, H., Zarrati, A. R., Jabbari, E., & Tavakkol, S. (2013). Velocity field in a reservoir in the presence of an air-core vortex. *Proceedings of the Institution of Civil Engineers, Water Management*, 356-364.
- Versteeg, H. K., & Malalasekera, W. (2007). Computational Fluid Dynamics, An introduction to the Finite Volume Method (2nd Edition). Pearson Education Limited, Edinburgh.