TECHNICAL TRANSACTIONS | CZASOPISMO TECHNICZNE
ENVIRONMENT ENGINEERING | ŚRODOWISKO

1-Ś/2016

DOI: 10.4467/2353737XCT.16.191.5940

RAJA ABOU ACKL, PAUL UWE THAMSEN*

EXPERIMENTAL AND NUMERICAL INVESTIGATIONS ON AIR ENTRAINMENT IN PUMP SUMP FOR WET PIT PUMPING STATIONS

BADANIA EKSPERYMENTALNE I NUMERYCZNE NAD PORYWANIEM POWIETRZA W STUDZIENCE ŚCIEKOWEJ DLA STACJI POMP MOKRYCH

Abstract

This paper presents a detailed study carried out in order to predict the influence of various design parameters and operating conditions on the performance of the wet pit pumping station. The method is based on the use of physical and numerical models to reproduce and understand the flow conditions inside the wet pit and their effect on the air entrainment. The research starts with developing a representative model for the wet pit pumping station, goes through conducting the numerical and experimental studies as well as the validation of the results, and ends with providing a valuable easy-to-enforce preliminary design and operating recommendations.

Keywords: air entrainment, CFD, pumping station wet pit

Streszczenie

W niniejszym artykule przedstawiono szczegółowe badania przeprowadzone w celu przewidywania wpływu różnych parametrów projektowych i warunków pracy na działanie stacji pomp mokrych. Metoda ta opiera się na wykorzystaniu wzorów fizycznych i numerycznych w celu odtworzenia i zrozumienia warunków przepływu wewnątrz pompy mokrej oraz ich wpływu na porywanie powierza. Badanie rozpoczyna się od opracowania reprezentatywnego modelu stacji pomp mokrych, następnie skupia się na przeprowadzeniu badań eksperymentalnych i numerycznych oraz walidacji wyników, kończąc na przedstawieniu cennych i łatwych do wdrożenia rekomendacji dotyczących wykonania projektu wstępnego i eksploatacji.

Słowa kluczowe: porywanie powietrza, CFD, stacja pomp, pompa mokra

---

[*] M.Sc. Raja Abou Ackl, Prof. D.Sc. Ph.D. Eng. Paul Uwe Thamsen, Chair of of Fluid System Dynamics, Department of Fluid Dynamics and Technical Acoustics, Technische Universität Berlin.

1. Introduction

In many places, lifting systems represent central components of a wastewater system. Pumping stations with circular wet-pit design are the most popular for smaller pump stations due to their relatively simple construction techniques as well as a smaller footprint for a given sump volume [1].

This type of pumping stations is equipped with submersible pumps located, in this case, directly in the wastewater collection pit. The wastewater passes through the pump station untreated and loaded with all kind of solids. Thus, the role of the pump sump is to provide an optimal operation environment for the pumps in addition to the transportation of the sewage solids. Bad design of the pit may affect the overall performance of the station in terms of poor flow conditions inside the pit, non-uniform and disturbed inflow at the pump inlet, as well as air entrainment to the pump. Therefore, understanding the effects of design criteria concerning pumping station performance is important in order to fulfil the wastewater transport maintenance-free and as energy efficient as possible.

The aim of this article is to explain a method evaluating the impact of various design and operating conditions on the performance of the pump station concerning the air entrainment to the pump as well as inside the pit.

2. Air entrainment in wet pit pumping stations

Due to the 'intermittently' working conditions of the wet pit pumping stations, the height between the inlet pipe and the free liquid surface in the pit changes constantly. This means that the wastewater forms a free jet falling on to the sump wastewater surface (Fig. 1). Depending on the velocity and the height of the falling jet, air bubbles may be entrained beneath the surface by the plunging jet [2–4].

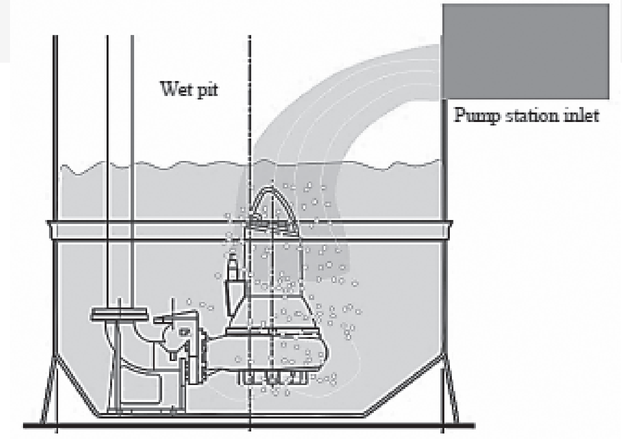

Fig. 1. Typical air entrainment in wet well pumping station

The air entrainment by plunging jets takes place when the jet impact velocity exceeds a critical value, which is a function of the inflow conditions [2–3]. The maximum penetration depth of the air bubbles depends on many parameters, such as the impact diameter of the jet, the velocity at the water surface, the jet instability and the free surface deformation [4, 6]. Therefore, it is difficult to predict theoretically. Entrained air caused by the impacting jet has an influence on the liquid flow field and on the debris transport in the sump [5]. Moreover, the performance of the pump will be affected if the air bubbles enter it. In general, centrifugal pumps can pump water with up to 5–10% gas content [7]. Yet, lower amounts of gas already influence the pumping system operation, changing the power consumption and hydraulic head, which reduce the efficiency of the system. Furthermore, vibrations arise that will damage the bearings. Thus, air entrainment of all amounts should be avoided and be considered during the design process.

Most of the studies on the mechanism of air entrainment and the penetration depth of a plunging jet are based on experimental investigations for jets from nozzles falling into a receiving reservoir or for a velocity that is not applicable in the wastewater system, see example publications [2, 6, 8, 18]. Moreover, the flow inside the pump sumps is irregular and very complicated with three dimensional patterns, due to the flow induced by the suction effect of the working pumps, in addition to the interaction between the tight space and the flow in the sump.

Thus, the conclusions and the formulas derived from these studies are not, or at least partly, applicable in the case of an impinging jet into a wet pit pumping station.

3. Methodology

The ability to predict the flow conditions and the air entrainment in the sump is very important during the design process in order to improve the design and avoid the air entrainment to the pump. The physical laboratory study is a precious tool that helps to describe hydraulic phenomenon and predict very complicated flow conditions. However, it is common to use a scaled physical model in the laboratory due to economical and spatial demands. Due to hygienic reasons, water is used to imitate the wastewater in this stage of research.

3.1. Scale effects

A physical model is a geometrically reduced or sometimes enlarged reproduction of a real-world prototype [12], and is used as a research tool for finding the technically and economically optimal solution of engineering problems [9]. In free-surface flows, which this case is, gravity plays an important role and thus Froude similitude should be used in order to fulfil the geometric similarity of the water surface [9]. This similarity means that the Froude number, as shown in Eq. (1), is kept identical both in the model and the prototype.

$$\mathrm{Fr} = \left(\frac{\text{Inertial force}}{\textit{Gravity Force}}\right)^{0.5} = \frac{\mathrm{V}}{(gL)^{0.5}} \tag{1}$$

The entrainment of air bubbles by a plunging jet is governed by the surface tension and the flow turbulence, implying the use of Weber similitude and Reynolds similarity respectively. Thus, the air entrainment in the small model based upon Froude similitude could be affected, due to the underestimation of the flow turbulence and the overestimation of the surface tension. There are many countermeasures to minimize scale effects in Froude models, such as calibration, the replacement of fluid, the use of scale series and the use of limiting criteria concerning the force ratios [9, 11].

On the contrary, the numerical models represent the real problem with no need of scaling. Hereof, the idea arises to use the numerical simulation to predict the air entrainment in such complicated situations. But on the other hand, the simplifications in the numerical solution lead to some deviations between the model and the prototype. Therefor, there is a need to calibrate and assess the accuracy of the numerical simulation with experimental investigations.

3.2. Qualitative evaluation of the air entrainment

After Bin [6], the bubbles resulting from plunging liquid jet will be dispersed beneath the liquid surface and form two regions classified according to the size of the bubbles:

- The biphasic conical region containing bubbles that reach maximum depth where the buoyancy forces balance the momentum of the jet.
- The region of bigger rising bubble.

The prediction of the bubble size is difficult. In addition, the accurate calculation and measurement of the air flow rate that enters the water is very complicated. However, as already mentioned above, air entrainment of all amounts should be avoided, and consequently, it is irrelevant to measure and find the exact amount of air entrained to the sump, but rather to find the operation and flow conditions that led to air entrainment into the pump.

In this research, two methods are used to evaluate the air entrainment in the pumping station.

1. Measuring the dimension of the bubble cloud (Fig. 2) i.e. the region occupied with the bubbles, which represents a qualitative evaluation of the air entrainment in the sump.
2. Observing the air bubbles in the outlet pipe of the pump. This gives an indicator of the air entrainment to the pump, which is the most important one.

In this sense, a physical model that fulfils the demands of these methods is designed.

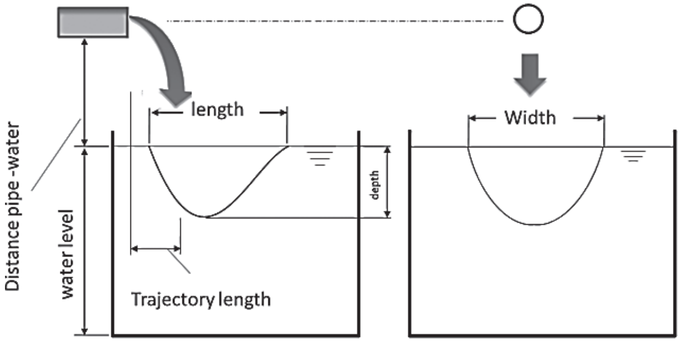

Fig. 2. Sketch of the 'entrained bubble cloud'

3.3. The physical test model

The physical model is designed upon Froude similarity. The scale 1:3.2 is chosen to simulate a typical market standard sump of a 1600 mm diameter with a standard acrylic cylinder of 500 mm. The height of the model is 750 mm and the inflow pipes are mounted at a height of 450 mm above the bottom of the tank, reproducing the height of 1440 mm in original full-scale design.

A simplified model has been used to represent the most important features of the duplex circular wet pit pumping station. The Representative Model consists of: coupling systems, guide bars, pressure pipe and dummy pumps. The need of reference geometry implies the use of a simple tank with a flat floor, called the ‘baseline geometry’. The inflow direction relative to the pumps centreline position as well as the water level can be changed in the model (Fig. 3).

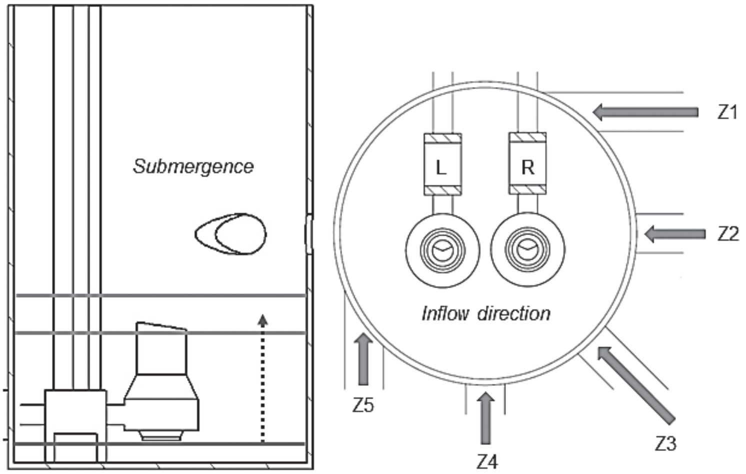

Fig. 3. The adaptable parameters of the model

The implemented test rig of the physical model is made of acrylic (Fig. 4) to enable the observation of the aerated region inside the sump and the bubbles entering the dummy pump at various operating conditions. It is possible to conduct the experiments in two working conditions:

1. Varied water level to reproduce pumping cycles.
2. Constant water level to enable the conduction of the test for constant inflow.

3.4. Numerical test model

There are many numerical simulation studies concerning the air entrainment, for example [4] presents 2D simulation the results of round vertical liquid jet plunging into bath with focus on bubbles size and the influence of the jet velocity using smoothed volume of flow technique [20]

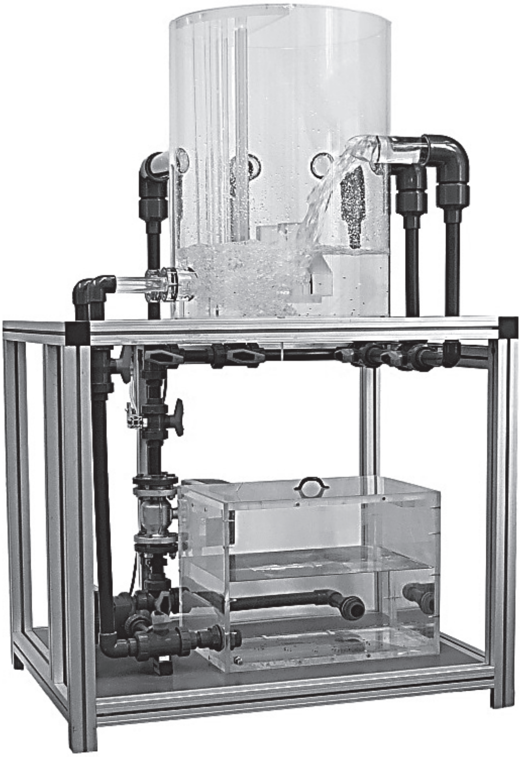

Fig. 4. Model test rig

studied the formation of the air cavities generated by translating plunging jet into a pool, whereas [5] researched the influence of the air entrainment on fibber transportation in the sump.

In our study, the Volume of Fluid (VOF) method was used to simulate the air entrainment and the air bubble formation. The water and the air are assumed to be incompressible Newtonian fluids. In this method, a phase q_A of the multiphase fluid consisting of the phases q_W (water) with the density ρ_W and q_A (air) with the density ρ_A is described by its volume fraction (α_A) in a computational cell. For α_A the following three states apply:

$\alpha_A = 0$, the cell is empty of the q_A phase,

$\alpha_A = 1$, the cell is full of the q_A phase,

$0 < \alpha_A < 1$, the cell contains the interface between the q_W and the q_A phase.

In the absence of sources of mass and momentum, the continuity equation for the volume fraction of the phase q_A is written as:

$$\frac{\partial}{\partial t}\left(\alpha_q \rho_A\right) + \nabla\left(\alpha_q \rho_A u\right) = 0 \tag{2}$$

Thereupon, the q_W phase volume fraction is computed from the relation: $\alpha_W + \alpha_A = 1$. The momentum equation depends on the volume fractions of the phases through the properties ρ and μ.

$$\frac{\partial(\rho\mathbf{u})}{\partial t} + \nabla(\rho\mathbf{uu}) = -\nabla p + \nabla\left[\mu\left(\nabla\mathbf{u} + \nabla\mathbf{u}^T\right)\right] + \rho g + F \tag{3}$$

In this equation **u** is the velocity vector, ρ is the density, p is the pressure, g is acceleration of gravity and F is the equivalent volume force due to the surface tension.

A transport equation is solved for the water phase to model the surface between water and air in the absence of any inter-phase mass transfer:

$$\frac{\partial}{\partial t}(\alpha_w) + \nabla(\alpha_w u) = 0 \tag{4}$$

In this two-phase system, the volume fraction of the air phase is being tracked, the density in each cell is given by:

$$\rho = \alpha_A\rho_A + (1 - \alpha_A)\rho_W \tag{5}$$

The viscosity μ is computed in the same manner:

$$\mu = \alpha_A\mu_A + (1 - \alpha_A)\mu_W \tag{6}$$

Furthermore, k-e turbulence model was used with wall functions.

3.4.1. Computational domain and boundary conditions

Unsteady CFD calculations applying the CFD code ANSYS/Fluent were performed. Fig. 5 shows the 3D computational domain. It is discretised into 4.9 × 106 tetrahedral cells with max 3 mm length in each direction, and partitioned into 14 subdomains, every domain calculated with one processor. The assessment of the independence of the results on the computational grid was checked by further calculations, applying a finer grid to capture the smallest air bubbles, and the suitable mesh where selected, not presented here.

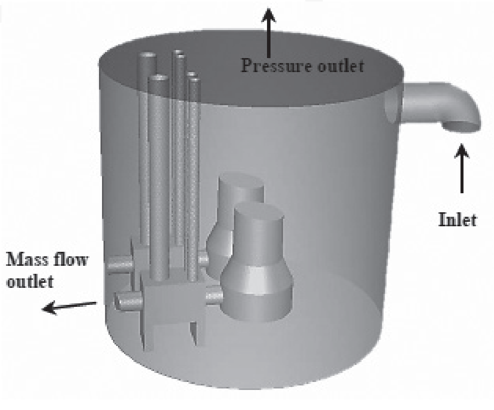

Fig. 5. Numerical setup

The calculations were made with the densities and viscosities of air and water at 15°C. The simulations were performed with a water-air surface tension set at 0.073 N/m. The PISO algorithm was used for pressure–velocity coupling. The interface between fluids was represented with Geometric Reconstruction Scheme. At the inlet, a velocity profile is specified for the velocity. The open top of the pit has a pressure outlet boundary condition, and the outlet of the simulation geometry has a mass flow boundary condition. With this combination, the water level in the pit remains constant. Computations were continued until the global residuals reached 10−5. During the unsteady calculation, an adaptive time-stepping method was used to ensure a Courant-Friedrichs-Levy number less than 1. At the initial time, the domain is filled up to a certain level with water. The remaining region consists of air.

4. Results and conclusions

The experimental investigations were systemically conducted by choosing an inflow direction and carrying out the experiments at various water levels and flow rates. During every experiment, the bubbles zone was monitored using video recording. Moreover, the acrylic outlet pipes of the dummy pumps were observed for air bubbles (Fig. 6). Depending on the records of the depth, the width and the length of the cloud were determined.

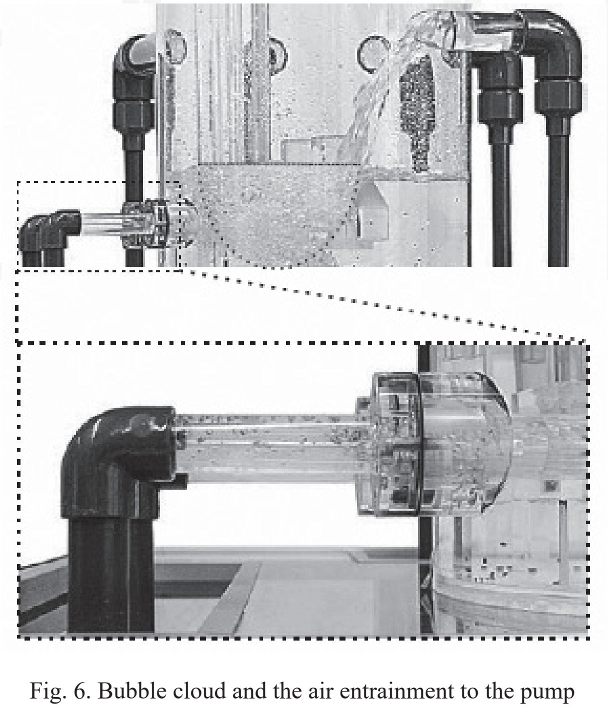

Fig. 6. Bubble cloud and the air entrainment to the pump

A series of computations was started with boundary conditions that represent the experimental flow conditions. The simulated time was 30 s. After that, a video was generated and the same procedure of the experimental investigation was applied, concerning the dimensions of the bubbled region (Fig. 7).

The air entrainment to the pump was analysed by checking the appearance of the air phase using a control surface positioned on the suction side of the pump. The number of the executed experimental test series was very large, as a result of the diversity of the working combinations for five chosen water levels (Fig. 8).

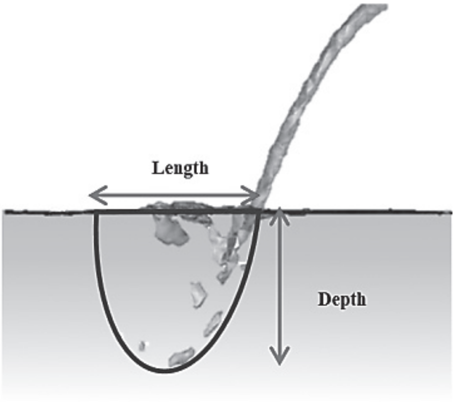

Fig. 7. Measured variables of the air entrainment

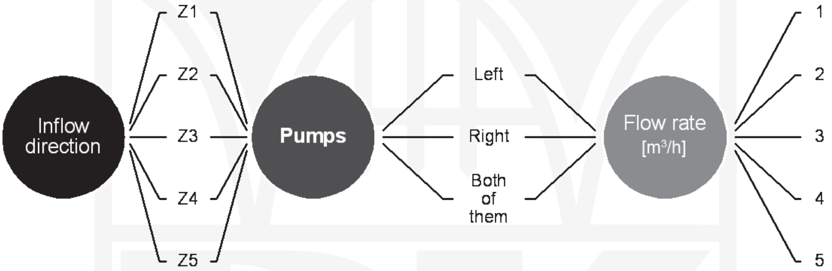

Fig. 8. Possible working combinations

Thus, only 8 cases were simulated in the numerical model, namely the inflow direction Z2 at two water level and four flow rates as shown in the Table 1.

Table 1

Summary of the numerical cases

Water level [m]	0.2	0.32
Flow Rate [m^3/ h]	1	1
	2	2
	3	3
	4	4

Both experimental and numerical results are presented. The experimental observations were used to assess the impact of the inflow direction on the air entrainment to the pump and to validate the simulation results regarding the dimensions of the bubble cloud and the (yes/no) indicator of the air entrainment to the pump.

4.1. The minimum submergence required to avoid air entrainment

Fig. 9 shows the experimental results of the minimum submergence for all possible combinations of inflow direction, flow rate and the working pump, in addition to a schematic illustration of the inflow direction influences on the pumps inside the pit. The minimum submergence is the water level required to prevent air entrainment to the pumps, which means, in the experiments, if the water level beneath this submergence, bubbles appear in the outlet pipe.

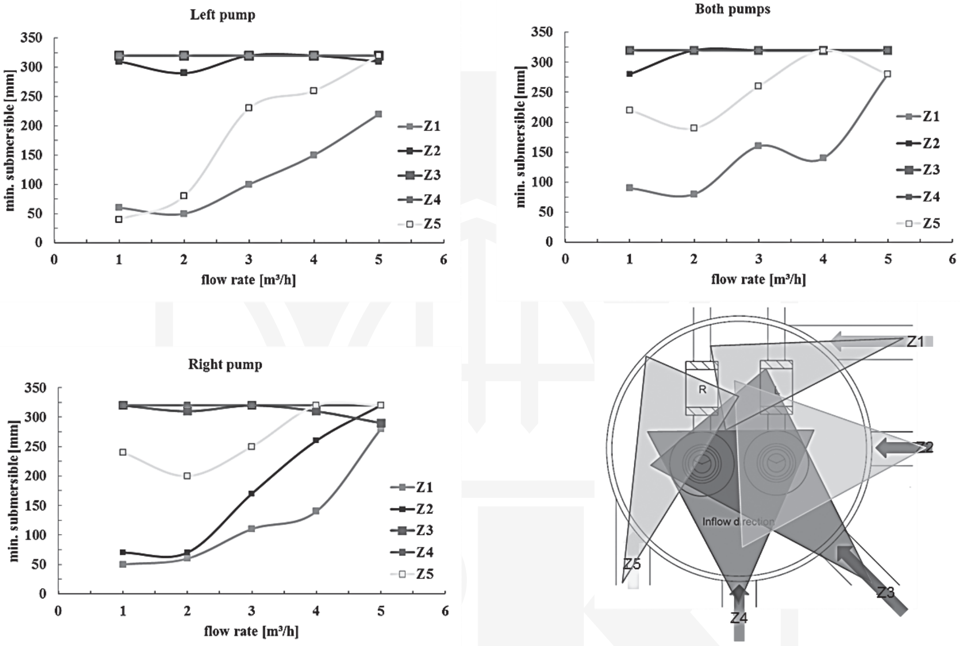

Fig. 9. Experimental results for minimum required submergence at different working combinations

The experimental results show that the flow directions Z3 and Z4 need the highest water level to prevent the air bubbles from reaching the pumps. Whereas it was very clear that both tangential directions produce suitable flow conditions inside the pumping station that minimize the amount of entrained air to both pumps.

Therefore, it can be concluded that the best inflow direction to the pump station concerning the air entrainment is the Z1 because it is associated with the lower water level.

4.2. Dimensions of the bubbles cloud

By determining the dimensions of the cloud, the average bubble size is considered. From Fig. 10, it can be seen that the simulation results and the experimental values are relatively in good agreement. The behaviour is captured and the results reveal that the model and the interface tracking are able to capture the bubble cloud generated from an impinging jet.

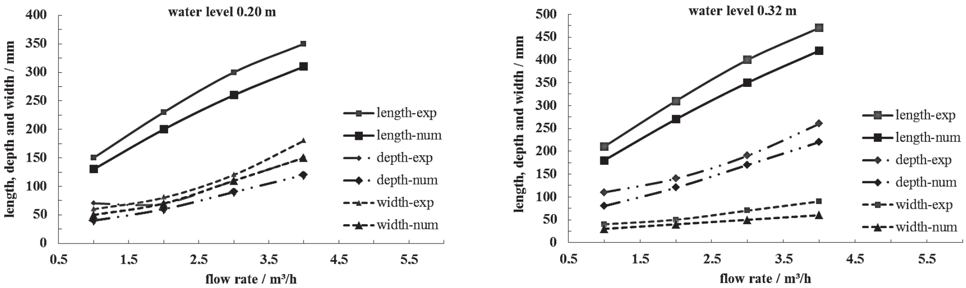

Fig. 10. Experimental and numerical dimensions of the bubbles cloud

The results show that the numerical model underestimates the phenomena. The deviation can be explained by the uncertainties of the numerical model and the inaccuracy of the measurements in the experimental setup.

At the inlet, the turbulence is neglected, which means that the jet instability and the free surface deformation are not sufficiently reproduced. The air entrainment is highly affected by the turbulence in the jet and the free surface of the receiving pool.

Furthermore, the measurements of the flow rate, the dimensions of the bubble zone as well as the water level contain some inaccuracy, leading to a mismatch between the boundary conditions in both setups and to deviations in the results.

4.3. Air entrainment to the pump

The air entrainment to the pump depends on the balance of various forces that act on the bubbles, depending on their volume, namely the buoyancy forces, the momentum of the moving bubbles and the effect of the flow induced from the working pump. Considering all the difficulty to capture small bubbles, the dimensions of the bubble cloud are underestimated. In this sense, it can be justified that the numerical model succeeded to predict the air entrainment to the pump in 5 of the 8 cases.

However, the calculations of the rest of the cases are planned, which will give a better indicator on the accurate of the numerical simulation.

5. Summary

The presented paper described the research of experimental and simulation investigations to assess the influence of inflow direction on air entrainment from a plunging water jet on a free surface of a wet pit pumping station. The experimental investigations conducted on the model of a wet pit pumping station show the influence of inflow direction and the water level on the performance of the station with respect to air entrainment. As a conclusion derived from the model test, the use of the tangential inflow direction can be suggested. The three-dimensional numerical model was applied to reproduce the experimental setup in order to predict the air entrainment and the flow characteristics at several flow conditions. The air entrainment was qualitatively evaluated by defining the dimensions of the bubble cloud and observing the air bubbles entering the suction side of the pump. The numerical VOF model proved to give relatively acceptable results compared to the experimental data. The dimensions of the cloud in the numerical approach had the same behaviour of the experimental model, but with some underestimations. Assessing the use of CFD as tool in the scale series is planned as future works to define the scale effect on the air entrainment and evaluate the transferability of the model results to the original prototype.

References

[1] American National Standards Institute ANSI/HI 9.8, 2012, *American National Standard for Pump Intake Design*.
[2] Zhu Y., Oguz H.N., Prosperetti A., *On the mechanism of air entrainment by liquid jets at a free surface*, Journel of Fluid Mechanics, Vol. 401, 2000, 151–177.
[3] Kiger K.T., Duncan J.H., *Air-Entrainment Mechanisms in Plunging Jets and Breaking Waves*, Annual Review of Fluid Mechanics, Vol. 44, 2011, 563–596.
[4] Qu X.L. et al., *Characterization of plunging liquid jets: A combined experimental and numerical investigation*, International Journal of Multiphase Flow, 37, 2011, 722–731.
[5] Krepper E. et al., *Influence of air entrainment on the liquid flow field caused by a plunging jet and consequences for fibre deposition*, Nuclear Engineering and Design 241, 2011, 1047–1054.
[6] Bin A.K., *Gas entrainment by plunging liquid jets*, Chemical Engineering Science, Vol. 48, 1993, 3585–3630.
[7] Gülich J., *Kreiselpumpen: Handbuch für Entwicklung, Anlagenplanung und Betrieb: Ein Handbuch für Entwicklung, Anlagenplanung und Betrieb*, Springer, 2008.
[8] Chanson H. et al., *Similitude of Air Entrainment at Vertical Circular Plunging Jets*, Proceedings of ASME FEDSM'02, 2002.
[9] Heller V., *Scale effects in physical hydraulic engineering models*, Journal of Hydraulic Research, Vol. 49, No. 3, 2011, 293–306.
[10] Weismann D., Gutzeit T., *Kommunale Abwasser-pumpwerke*, Vulkan Verlag, 2006.
[11] Bollrich D., *Technische Hydromechanik. 2. spezielle Probleme*, Verl. für Bauwesen 1989.
[12] Kobus H., *Wasserbauliches Versuchswesen. DVWK Schriften 39. Spezielle Probleme*, Verl. Paul Parey 1984.
[13] Kawakita K. et. al., *Experimental study on the similarity of flow in pump sump models*, 26th IAHR Symposium on Hydraulic Machinery and Systems, 2012.
[14] Padamabhan M. et. al., *Scale Effects in Pump Sump Models*, Journal of Hydraulic Engineering, Vol. 110, 1984, 1540–1556.
[15] Chanson H., Aoki S., Hoque A., *Bubble entrainment and dispersion in plunging jet flows: freshwater versus seawater*, J Coastal Res, 22 (3), 2006, 664–677.
[16] Chanson H., *Turbulent air-water flows in hydraulic structures: dynamic similarity and scale effects*, Environ Fluid Mech, 9 (2), 2008, 125–142.
[17] Nakasone H., *Study of Aeration at Weirs and Cascades*, Journal of Environmental Engineering, Vol. 113, No. 1, 1987, 64–81.

[18] Yamagiwa K., Mashima T., Kadota S., Ohkawa A., *Effect of liquid properties on gas entrainment behavior in a plunging liquid jet aeration system using inclined nozzles*, Journal of Chemical Engineering of Japan, 26, No. 3, 1993, 333–336.

[19] Sande E. van der, Smith J.M., *Eintragen von Luft in eine Flussigkeit durch einen Wasserstrahl*, Chemie-Ing.-Techn. 44, Jahrg. 1972/Nr. 20.

[20] Brouilliot D., Lubin P., *Numerical simulations of air entrainment in a plunging jet of liquid*, Journal of Fluids and Structures Volume 43, November 2013, 428–440.


PK