Jun 25th, 12:00 AM - Jun 27th, 12:00 AM

Three-Dimensional Numerical Modeling of a Scroll Vortex Intake

S. N. Chan
The Hong Kong University of Science and Technology, treechansn@ust.hk

7th International Junior Researcher and Engineer Workshop on Hydraulic Structures, IJREWHS'19, B. HEINER and B. TULLIS (Eds), Report 1, U.S. Bureau of Reclamation, Denver, CO, USA - ISBN 978-0-578-69809-0
https://doi.org/10.26077/5qw4-ne72

THREE-DIMENSIONAL NUMERICAL MODELING OF A SCROLL VORTEX INTAKE

S.N. CHAN
Department of Civil and Environmental Engineering and Institute for Advanced Study,
The Hong Kong University of Science and Technology, Hong Kong, China, treechansn@ust.hk

Abstract: Scroll vortex intakes are vortex drop structures commonly used in water supply, drainage and sewerage systems, characterized by a vortex chamber with its wall curling inwards to the dropshaft and a horizontal bottom. The stormwater flows into the intake via an eccentrical approach channel, which imparts vortex motion to the flow, forming a swirling vortex flow with a stable air core through the center of dropshaft. Over past decades, much effort has been devoted to investigating the scroll intake vortex flow, yet the understanding and predictions of the vortex flow is still far from complete due to a lack of detailed investigation on its velocity field and air core characteristics. In this work, a three-dimensional (3D) computational fluid dynamics (CFD) model using the Volume-of-Fluid (VOF) method is used for investigating the complex flow of a scroll vortex intake. The CFD model predictions are validated with detailed flow profile, flow velocity and air core measurements on a physical hydraulic model. It is found that the vortex flow in the scroll chamber resembles a free vortex and the circulation is approximately equal to that at the inlet to the chamber, with a thin bottom boundary layer. For the vortex flow at the bellmouth outlet, the tangential velocity distribution satisfies a Rankine vortex. Furthermore, the vortex flow at the bellmouth outlet possesses a circulation constant which is smaller than that in the chamber.

Keywords: scroll vortex intake, swirling flow, air-core, Volume-of-Fluid method.

INTRODUCTION

Scroll vortex intakes are vortex drop structures used in water supply, drainage and sewerage systems. A scroll vortex intake (c.f. Fig. 1) is characterized by a vortex chamber with its wall curling inwards to the dropshaft and a horizontal bottom. The water flows into the intake via an eccentrical approach channel. Thus the intake can impart the vortex motion to the flow and then a swirling vortex flow with a stable air core is formed along the dropshaft. The air core acts as a chimney for the air trapped into the approaching flow to escape freely, and the extended path of water motion increases the energy dissipation of the flow. Therefore, such vortex intakes usually possess the hydraulic characteristics of low air entrainment and high energy dissipation.

The concept of scroll intake design was first introduced by DRIOLI (1947). Since then, much effort has been devoted to predict the scroll intake vortex hydraulic characteristics. Past investigation

heavily depends on physical hydraulic models and the idealized vortex flow pattern for modeling the flow process (JAIN and KENNEDY, 1983; LEE et al. 2006; MULLIGAN et al. 2016). A standard design of scroll intake geometry has been proposed by DRIOLI (1969) and JAIN and ETTEMA (1987). A deeper understanding of the vortex flow process in scroll intakes promotes the attempts for reaching a rational approach for scroll intake design. VIPARELLI (1950) proposed a semi-empirical approach for predicting the head-discharge relationship of scroll intake. ACKERS and CRUMP (1960) proposed an analytical approach for scroll intakes by assuming the vortex flow satisfies a free vortex model. PICA (1970) proposed an improved semi-empirical solution that predicts the head-discharge relationship, later simplified into a dimensionless form by HAGER (1985). However, the prediction of the air core size by many of these theoretical models fails to agree well with experiment and field observation data. The scarce data of the detailed vortex flow field in literature make it difficult to develop a more accurate rational model for hydraulic design of scroll intakes. GUO (2012) measured, for the first time, the detailed flow field of a typical scroll intake design using Laser Doppler Anemometry (LDA). Based on the experimental data, the theory of ACKERS and CRUMP (1960) is considered the most justified, and an improved model is proposed.

Attempts have been made to model the air-water flow in vortex intake problems using three-dimensional (3D) computational fluid dynamics (CFD) models with the Volume-of-Fluid (VOF) method. Using VOF, the interface between the water and air can be located and tracked when it moves through the computational domain. CHAN et al. (2018) used a 3D CFD model to study the detailed flow structures of a stable tangential vortex intake. The model prediction was validated with measurements of head-discharge, air core and velocity profiles, which sheds light on the physics of a tangential vortex intake. An empirical relationship was also established for predicting air core size according to the flow and geometry of the intake.

This paper presents the 3D CFD modeling results of a scroll vortex intake of typical design. The numerical model results are validated against flow profile, velocity and air core measurements on a physical model of a scroll vortex intake. In this paper, the details of the CFD model are first presented; the numerical model results are then discussed and compared with experimental measurements.

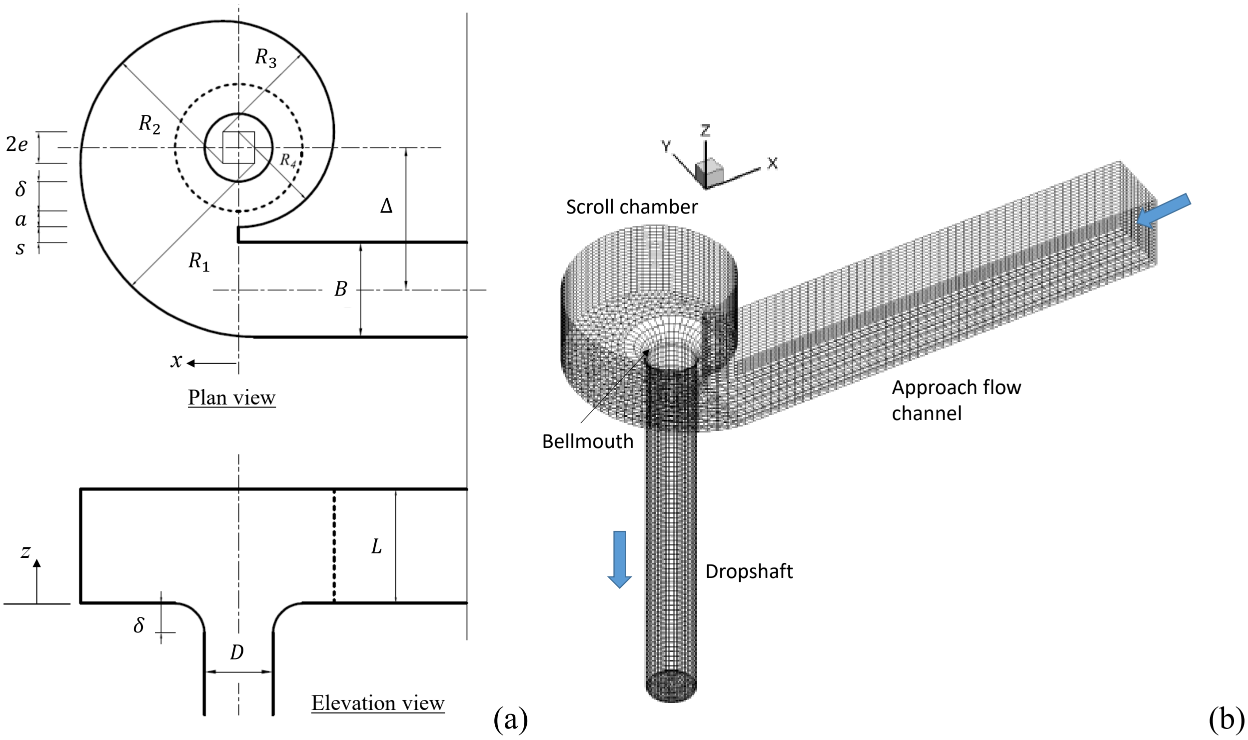
COMPUTATIONAL FLUID DYNAMICS MODEL

The scroll vortex intake model investigated in this study is designed based on the standard geometry recommended by DRIOLI (1969). Fig. 1a schematically shows the geometry and key dimensions of the scroll vortex chamber. The chamber wall consists of four arcs with decreasing radii in the flow direction. The center of each arc is offset from the dropshaft center by a certain distance, which is determined by an eccentricity $e = (B + s)/7$, where $B = 0.167$ m, approach channel width, and $s = 27$ mm, offset distance between the approach channel wall and the chamber wall. The radius of each arc, R_1 to R_4, is then determined by following relations (Eq. 1):

$$R_4 = D/2 + \delta + a + e,\ R_3 = R_4 + e,\ R_2 = R_3 + 2e,\ R_1 = R_2 + 2e,\ \Delta = R_1 + e - B/2 \qquad (1)$$

In Eq. (1), the dropshaft diameter, $D = 0.12$ m; radius of curvature of the dropshaft entrance, $\delta = 0.052$ m; distance between the top of bellmouth and the chamber wall, $a = 0.028$ m; and the eccentricity between the approach channel and the dropshaft, $\Delta = 0.25$ m. B and D are first determined based on the design discharge. Then Δ, s and a are chosen based on structural considerations. Finally e and R_1 to R_4 are determined based on Eq. (1). Experimental measurements of flow profile, velocity field and air core sizes of such scroll vortex intake design has been reported by LEE et al. (2006) and GUO (2012).

Fig. 1 – (a) Geometry definition of a scroll vortex intake, (b) 3D view of the computational mesh.



The VOF model (HIRT and NICHOLS, 1981) predicts water flows with a free surface by solving a single set of momentum equations and tracking the volume fraction of water and air throughout the domain. The tracking of the interface between the phases is accomplished by the solution of the transport equation for the volume fraction of water phase α_w (Eq. 2):

$$\frac{\partial}{\partial t}(\alpha_w \rho_w) + \nabla \cdot (\alpha_w \rho_w \mathbf{U}) = 0 \qquad (2)$$

A single momentum equation is solved throughout the domain, and the resulting phase-averaged velocity field $\mathbf{U} = (u, v, w)$ is shared among the phases (Eq. 3).

$$\frac{\partial}{\partial t}(\rho \mathbf{U}) + \nabla \cdot (\rho \mathbf{U}\mathbf{U}) = -\nabla P + \nabla \cdot [\mu_t (\nabla \mathbf{U} + \nabla \mathbf{U}^T)] + \rho \mathbf{g} \tag{3}$$

where the air-water mixture density, ρ = (1-α_w)ρ_a + $\alpha_w\rho_w$; μ_T is the dynamic molecular viscosity obtained from a *k*-*ε* turbulence model; *P* is the pressure; the gravitational acceleration vector, $\boldsymbol{g}$ = (0,0,-9.81) m/s^2; ρ_w and ρ_a are water and air densities, respectively. The air-water interface is defined at water volume fraction α_w = 0.5.

The governing equations (2) and (3), and the *k* and *ε* equations of the turbulence model were solved numerically in the CFD software of ANSYS FLUENT 15 (ANSYS INC., 2013). A second order upwind advection scheme was used for momentum and density, while a first order upwind advection scheme was used for *k* and *ε*. The volume fraction equation is spatially discretized using the Modified High Resolution Interface Capturing (HRIC) Scheme in FLUENT. Convergence for each time step was declared when the normalized residual is less than 10^{-4} for all variables.

An unstructured boundary-fitted model grid was used (Fig. 1b). The computational mesh has 69,944 grid cells with hexahedral cells for the approach channel and dropshaft, and triangular-prismatic cells for the scroll chamber with mesh refinement close to the dropshaft wall. The minimum grid size near the dropshaft wall region is 2 mm. Mesh convergence was tested using a fine mesh with 436,880 cells. The differences on the predicted approach flow depth, air core size and swirling velocity are less than 5%.

The computational model has three open boundaries: the inlet, the outlet of the vortex dropshaft, and the top atmospheric boundary. The upstream inflow of the approach channel is prescribed with the total water flow rate and a head discharge relationship at the inlet of the model measured experimentally. The upper boundary of the CFD model is prescribed with zero gauge pressure/atmospheric pressure. The dropshaft outlet is prescribed with zero gauge pressure. A roughness height of 0.01 mm is used for all wall boundaries.

Numerical simulations were carried out for five representative flow rates of *Q* = 2, 4, 6, 8 and 10 L/s. The supercritical flow in the approach channel and the vortex intake developed from the inlet. A time step of 0.001 s was used for the simulation; typically about 20 s of flow simulation were required for the vortex flow to reach steady state with a computation time of approximately 20 h.

RESULTS AND DISCUSSIONS

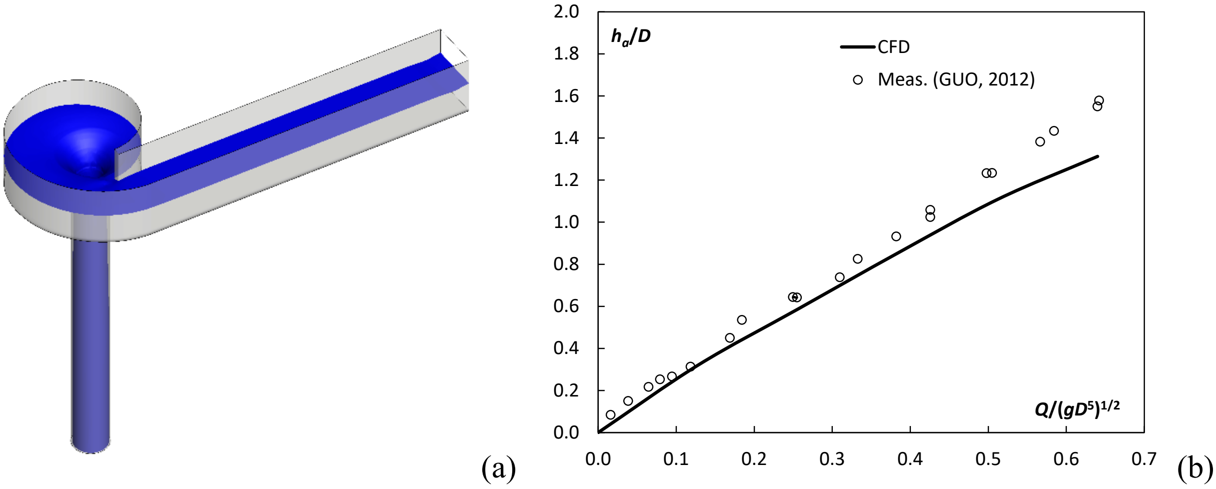
Flow-profile and head-discharge relation

Figure 2a shows the CFD predicted general vortex flow features for the case of *Q* = 6 L/s. It is seen that the flow surface is depressed as it approaches to the dropshaft. An air core is formed in the center

7th International Junior Researcher and Engineer Workshop on Hydraulic Structures, IJREWHS'19, B. HEINER and B. TULLIS (Eds), Report 1, U.S. Bureau of Reclamation, Denver, CO, USA - ISBN 978-0-578-69809-0
https://doi.org/10.26077/5qw4-ne72

and continues all the way along the dropshaft. After entering the dropshaft, the flow clings to the shaft wall and descends down the shaft wall vertically with a decreasing axis swirl. The flow depth in the approach channel and the scroll chamber increase smoothly with increasing discharge. No unstable feature is observed within this range of discharge (Q = 2-10 L/s). Figure 2b shows that the dimensionless approach flow depth h_a/D (measured at 70 mm upstream of the scroll chamber inlet using a point gauge) increases linearly with the dimensionless discharge $Q/(gD^5)^{1/2}$. The CFD prediction follows the trend of measured head-discharge relationship closely although it under-estimate the approach flow depth at high flows by a maximum relative error of 15%.

Fig. 2 – (a) Predicted free surface for Q = 6 L/s; (b) head-discharge relationship of the scroll vortex intake.



Vortex air core

Fig. 3a shows the predicted air core at the throat for Q = 10 L/s. The air-core is not axisymmetric; it has a larger flow thickness at the quadrant between the azimuth angles of 0º and 270º. The minimum air core (throat) lies between the bellmouth and the bottom of the scroll chamber. The minimum air core size along the vertical direction of the intake A_m decreases with increase in discharge. The predicted minimum air core size ratio $\lambda_m = A_m/(0.25\pi D^2)$ compare satisfactorily with the measurement with a maximum relative error of about 30% at high flows (Fig. 3b).

Scroll chamber flow

Due to the eccentric geometry, the inflow from the approach channel imparts angular momentum and results in a swirling flow in the scroll chamber (Fig. 4a). The swirling flow drains down the dropshaft through the bell-mouth. Fig. 4b shows the tangential velocity distribution at the azimuth of 75° (c.f. Fig. 4a) for Q = 6 L/s. Generally, the distribution of tangential velocity v_t decreases with greater radial distance r from the center of chamber. The vortex circulation $\Gamma = v_t r$ reveals that the flow satisfies a free vortex: Γ is approximately constant over r and equals the circulation at inlet $\Gamma = v_A\Delta$ (where v_A

is the average approach flow velocity = Q/Bh_a), except near the chamber bottom and at very low discharges where the viscous effect becomes significant. The CFD predicted tangential velocity and vortex circulation have excellent agreement with the measurement (Fig. 4b).

Fig. 3 – (a) Predicted air core at the throat for Q = 10 L/s, (b) Variation of minimum air core ratio with discharge.

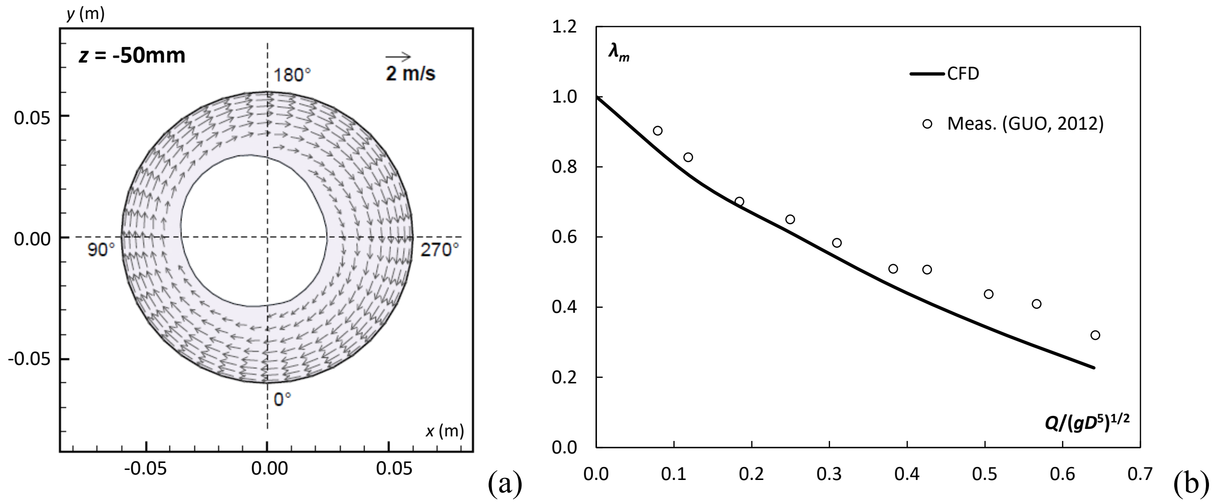
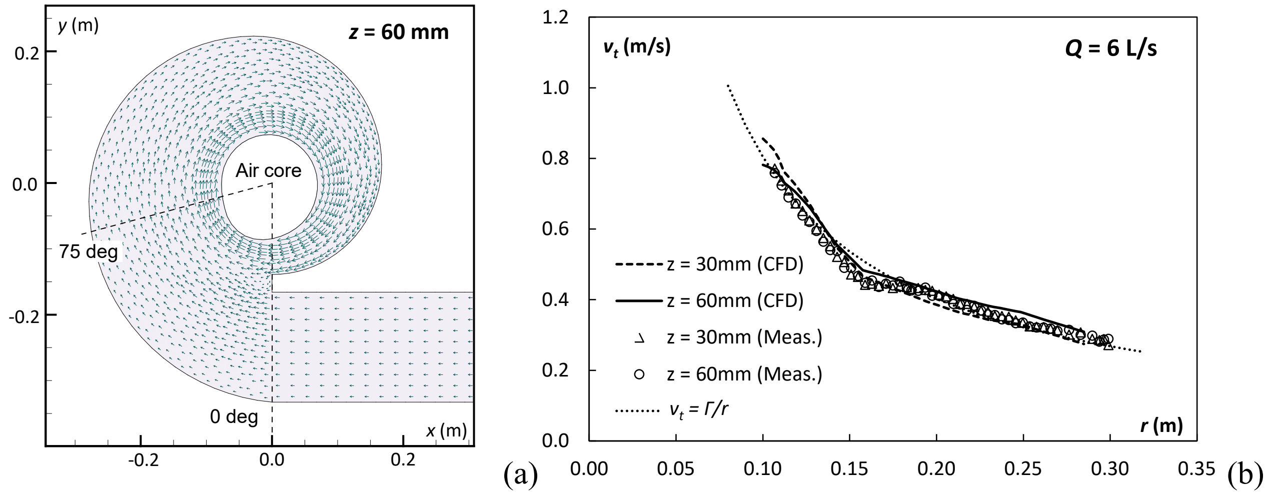


Fig. 4 – (a) Flow field and air core at the scroll chamber, (b) distribution of tangential velocity v_t at the scroll chamber of azimuth 75° for Q = 6 L/s, Γ = 0.08 m²/s. Measured data from GUO (2012).



Bellmouth flow

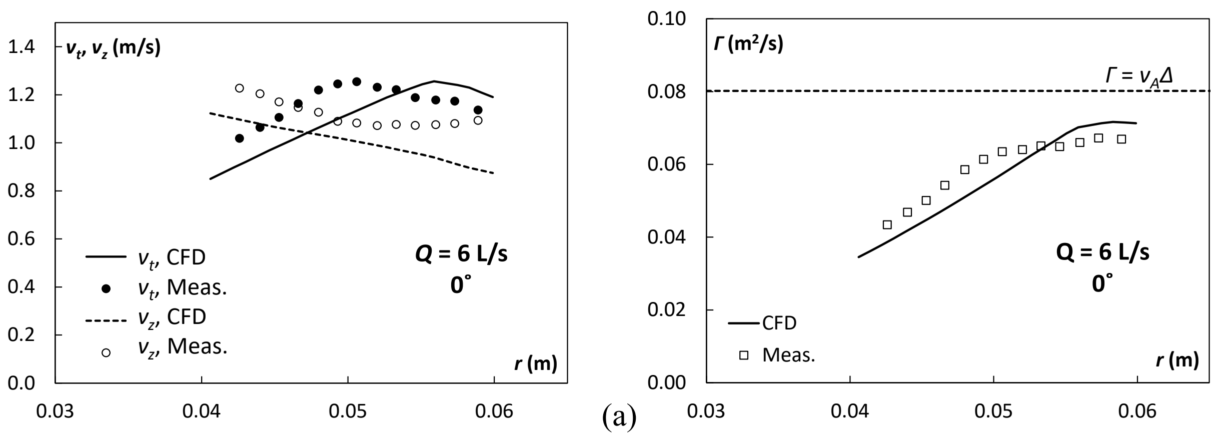
The existing scroll vortex theories consider the bellmouth as a control section, thus it is important to understand its velocity distribution to validate the assumptions of these theories. Fig. 5a shows the distribution of tangential velocity v_t and vertical velocity v_z of the bellmouth for Q = 6 L/s. In general, the flow can be divided into three regions: 1) a forced vortex region starting from the free surface of the air core to a certain radial distance within the vortex flow, where tangential velocity increases

7th International Junior Researcher and Engineer Workshop on Hydraulic Structures, IJREWHS'19, B. HEINER and B. TULLIS (Eds), Report 1, U.S. Bureau of Reclamation, Denver, CO, USA - ISBN 978-0-578-69809-0
https://doi.org/10.26077/5qw4-ne72

with greater radial distance from the shaft center; 2) a free vortex region in the remaining outside area where tangential velocity decreases; and 3) a boundary layer near the wall. This is similar to a Rankine combined vortex. The formation of the forced vortex region is due to the viscous dissipation at the air core region induced by the flow turbulence. It can be seen that the forced vortex region (increasing v_t) dominates the tangential flow field at the bellmouth (Fig. 5a).

In the forced vortex region, v_z decreases with increasing radial distance. This is consistent with the tangential velocity distribution of a Rankine combined vortex and the conservation of energy: the vertical velocity is constant for a free vortex flow (BINNIE and HOOKINGS, 1948). Since both tangential velocity and pressure increase with radius in the forced vortex flow, the vertical velocity decreases to maintain energy conservation. The prediction by CFD shows a relatively large region of forced vortex with increasing v_t, and a decreasing trend of vertical velocity against radial distance (Fig. 5a). The numerical prediction indicates that the assumptions of free vortex flow at the bellmouth by the model of ACKERS and CRUMP (1960) is not entirely correct. The vortex circulation, $\Gamma = v_t r$, shows an increasing trend with increasing radius (Fig. 5b). The maximum value of circulation is between 0.060 and 0.070 m^2/s, which is significantly lower than the initial circulation $\Gamma = 0.08$ m^2/s ($\Gamma = v_A \Delta$).

Fig. 5 – (a) Distribution of tangential velocity v_t and vertical v_z velocity, (b) distribution of vortex circulation at the bellmouth. Q = 6 L/s, azimuth angle of 0°. Measured data from GUO (2012).



(b)

CONCLUSIONS

In this study, the flow of a scroll intake vortex is studied using three-dimensional (3D) computational fluid dynamics (CFD) modeling. The head-discharge relationship and minimum air core sizes have been predicted by the CFD model. The minimum air core of the vortex flow occurs within the bellmouth. The vortex flow in the scroll chamber is similar to a free vortex with a circulation approximately equal to the circulation at the inlet, and the circulation gradually decreases as the flow swirls around the scroll chamber.

The predicted tangential velocity distribution of the vortex flow at the bellmouth is dominated by the forced vortex flow. The predicted vertical velocity distribution in the bellmouth decreases with increasing radial distance. The CFD modeling results provide useful insight for improving the current theoretical models for scroll vortex intake design.

ACKNOWLEDGMENTS

This study was supported by a grant from the Hong Kong University of Science and Technology (IGN17EG05).

REFERENCES

ANSYS INC. (2013). *ANSYS FLUENT 15.0 Theory Guide*.

ACKER, P., and CRUMP, E.S. (1960). "The vortex drop." *Proceedings of Institute of Civil Engineers, Part I*, 16(4), 433-442.

BINNIE, A.M., and HOOKINGS, G.A. (1948). "Laboratory experiments on whirlpools." *Proceedings of the Royal Society of London, Series A*, 194(1038), 398-415.

CHAN, S.N., QIAO, Q.S. and LEE, J.H.W. (2018). "On the three-dimensional flow of a stable tangential vortex intake." *Journal of Hydro-environment Research*, 21, 29-42.

DRIOLI, C. (1947). "Su un particolare tipo di imbocco per pozzi di scarico." *L'Energia Elettrica*, 24(10), 447–452 (in Italian).

DRIOLI, C. (1969). "Esperienze su installazioni con pozzo di scarico a vortice." *L'Energia Elettrica*, 46(6), 399–409 (in Italian).

GUO, J.-H. (2012). *Velocity field measurement of a scroll vortex intake flow*. M.Phil. Thesis, The University of Hong Kong.

HAGER, W. (1985). "Head-discharge relation for vortex shaft." *Journal of Hydraulic Engineering*, ASCE, 111(6), 1015–1020.

HIRT, C. W. and NICHOLS, B. D. (1981). "Volume of fluid (VOF) method for the dynamics of free boundaries." *Journal of Computational Physics*, 39(1), 201-225.

JAIN, S.C. and KENNEDY, J.F. (1983). *Vortex-Flow Drop Structures for the Milwaukee Metropolitan Sewerage District Inline Storage System*, Report No. 264, Iowa Institute of Hydraulic Research, The University of Iowa, Iowa City, Iowa.

JAIN, S.C., and ETTEMA, R. (1995). "Swirling flow problems at intakes." In: *Vortex Intakes, IAHR hydraulic structures design manual 1*, J. Knauss eds., Balkema, Rotterdam, The Netherlands, 125-137.

LEE, J.H.W., YU, D.Y. and CHOI, K.W. (2006). *Physical hydraulic model tests for Lai Chi Kok transfer scheme intake structure.* Croucher Laboratory of Environmental Hydraulics, The University of Hong Kong.

MULLIGAN, S., CASSERLY, J. and SHERLOCK, R. (2016). "Effects of geometry on strong free-surface vortices in sibcritical approach flows." *Journal of Hydraulic Engineering*, 142(11),

04016051.

PICA, M. (1970). "Scaricatori a vortice." *L'Energia Elettrica*, 47(4), 1–18 (in Italian).

VIPARELLI, M. (1950). "Su un particolare tipo di imbocco e sull'efflusso con vortice." *L'Energia Elettrica*, 27(10), 610–624 (in Italian).