Hydrodynamic Investigation of Free-Surface Turbulent Vortex Flows with Strong Circulation in a Vortex Chamber

S. Mulligan[1], J. Casserly[1] and R. Sherlock[2]
[1]School of Engineering and Design (CERIS)
Institute of Technology Sligo
Ash Lane, Sligo
IRELAND
[2]School of Science (CERIS)
Institute of Technology Sligo
Ash Lane, Sligo
IRELAND
e-mail: seanmulligan23@gmail.com

Abstract: *The results obtained from analytical, numerical and experimental modelling of free-surface vortex flows are presented. Vortex flow in an open channel flow chamber is simulated using the ANSYS CFX steady Eulerian-Eulerian multiphase flow model with various turbulence closure methods. The water surface and tangential velocity profile are also modelled using the Vatistas ($n = 2$) vortex model. These previous techniques are validated using particle tracking velocimetry data obtained from a physical model. Sensitivity analysis carried out on the numerical model presents a case for mesh independence and the results suggest that a structured mesh is essential. Curvature correction makes a significant improvement to the shear stress transport turbulence model. The Reynolds stress model produced the most accurate results; however, it suffers the drawback of computational expense. Errors in the numerical models were found to be in the region of 25% for the water surface and 17% for the tangential velocity. It is concluded that transient modelling is required to further improve the numerical simulation. Comparative results on the water surface and tangential velocity distribution signifies that the Vatistas $n = 2$ model agrees well with physical data.*

Keywords: *Vortex, Free-surface, CFX, Multiphase, Eulerian-Eulerian, PTV.*

1. INTRODUCTION

Strong, full air core vortices have been found to be beneficial when employed in various industrial systems such as vortex drop shafts (Drioli, 1947) confined vortex chambers (Vatistas, 1986) and vortex hydropower plants. This increasing demand for the use of strong vortex flows necessitates an improved understanding for the underlying hydrodynamics of this type of flow system. In recent years, numerical modelling has become a preferred form of analysis in industry. However, there are still only a few applications of multiphase analysis in vortex flows that exhibit sufficient validity. This is generally due to the complexity of the vortex near field due to anisotropic turbulence behaviour and the need for proper multiphase consideration of the free-surface interface. In this work, efforts are made to experimentally validate the ANSYS CFX code for a steady analysis of strong vortex flow behaviour.

Strong vortices are characterised by an intense axial flow at the discharge orifice in combination with a dominating tangential velocity field. The strength of vortex flow and the nature of the free-surface are strongly dependent on the pre-swirl activity of the approach flow circulation (Γ). The nature of the free vortex conserves angular momentum throughout the system which causes an inverse increase in tangential velocity radially in the majority of the flow field. Knowledge of the velocity fields and water surface profiles are of significant value to the hydraulic engineer for safe operation and quality control of devices. An ideal analytical solution for the tangential velocity in a viscous vortex was proposed by Rankine (1858) and later improved by Vatistas (1991) (discussed further in section 2). Efforts have been made to validate other analytical models through experimental methods (Anwar, 1965; Hite & Mih, 1994; and Wang et al 2011). In all cases, the investigations were carried out on weak air-core vortices above hydraulic intakes. Although the agreement between the theoretical and measured experimental velocity profiles were reported to be reasonable, the analytical models have a high degree of uncertainty when applied to real, complex three-dimensional fluid domains.

The approach of numerical modelling has become desirable in hydraulic engineering. Early work on numerical modelling of vortex flows is presented by Binnie & Davidson (1949) who applied a finite difference approximation to the free surface problem. Brocard et al. (1982) used a finite element modelling approach to calculate the circulation field in an intake using a criterion for the eddy viscosity. Trivellato (1995) performed finite volume method (FVM) modelling of free surface draining vortices in order to advance on his previous numerical investigation using the finite difference method. With commercial CFD codes becoming more readily available, numerous authors have presented benchmark work on intake vortices for various codes. Rajendran et al. (1999), Suerich-Gulick et al. (2006), Okamura et al. (2007), Hai-feng et al. (2008) and Tanweer et al. (2010) investigated free-surface vortices at intakes using a single phase model assuming that the water surface boundary condition is a frictionless free-slip wall. Advancement on this is presented by Bayeul-Lainé et al. (2012) who modelled the flow field as a multiphase system using two stratified fluid domains representing air and water independently. Suerich-Gulick et al. (2007), and Cristofano et al. (2014) carried out multiphase analysis on intake vortices employing the volume of fluid (VOF) method to capture the free-surface interface. Apart from the work carried out by Lei et al. (2011) very few multiphase models were implemented on vortex drop shafts that can be regarded as conclusive. It is noted (Suerich-Gulick, 2007) that most of the preceding work on the deformation of the free surface has only been modelled qualitatively and more quantitative work is needed. Regarding turbulence modelling, most of the previous simulations have been performed using the $k - \varepsilon$ turbulence model (e.g. Lei et al., 2011), and other RANS models which tend to overestimate turbulence (Tokyay & Constantinescu, 2005; Suerich-Gulich et al., 2006). Curvature correction for the RANS equations was developed (Spalart & Shur, 1997) but has not yet been tested for a wide range of vortex applications. Skerlavaj (2014) carried out an extensive review for a number turbulence models in a strong vortex accounting for curvature correction; however, this work was on a single phase case similar to before (Suerich-Gulick et al., 2007; Cristofano et al., 2014). Moreover, Reynolds stress modelling was not considered in this work.

It is recognised that little work on the validation of multiphase modelling of strong vortex flows has been carried out that present sufficient validity to be employed for industrial applications. Furthermore, previous literature failed to comment on the topic of mesh sensitivity in great detail. Moreover, the topic of turbulence modelling is not thoroughly investigated, particularly in strong vortex applications, and consequently this area of research requires further work. Therefore, the aim of this paper is to present a preliminary investigation of a steady, strong free surface vortex using the ANSYS CFX code and various mesh strategies and turbulence models to be validated using a physical model test case.

2. FLOW ANALYSIS

The flow is conveyed subcritically to the vortex chamber through a spiral inlet and discharges axially at some smaller radius (r_d) through the discharge orifice. The flow system is axisymmetric, three-dimensional and is described using the cylindrical co-ordinate system for tangential velocity (v_θ), radial velocity (v_r), and axial velocity (v_z) where their distributions are expressed as a function of the radius (r) from the vortex core origin. In the ideal vortex model, the flow is classified as irrotational and the tangential velocity profile takes the form of

$$v_\theta = \frac{\Gamma}{2\pi r} \qquad (1)$$

However, in a real Newtonian fluid, effects of viscosity limit the tendency of the fluid to achieve unrealistic velocities as $r \rightarrow 0$. To account for the viscosity in the near field, a simple analytical model was proposed by Rankine (1884) which describes the velocity field by representing the core region through solid body rotational $r < r_c$; where r_c is the core radius. This was later improved by Vatistas et al. (1991) who devised a model that overcomes the singular nature of Rankines model by using an algebraic family (represented by n) of velocity profiles highlighted in Eq. (2).

$$v_\theta = \frac{\Gamma}{2\pi}\left(\frac{r}{({r_c}^{2n} + r^{2n})^{\frac{1}{n}}}\right) \; when \; 0 \leq r \leq \infty \qquad (2)$$

where $n = 2$ produced god approximations for physical cases. By carrying out an order of magnitude analysis (neglecting viscosity) on the radial momentum equation of the Navier-Stokes equations and combining with Eq. (1) an expression can be arranged to describe the radial distribution of depth in the vortex as follows:

$$H_{\Delta} = \frac{1}{g}\left(\frac{\Gamma}{4\pi}\right)^2 \left(\frac{1}{r_{\mathrm{in}}^2} - \frac{1}{r_{\mathrm{r}}^2}\right) \quad (3)$$

where r_{in} *and* r_r are radii at the inlet and at an arbitrary radius in the vortex and H_{Δ} is the height difference between these radii. Both analytical expressions of Eq. (2) and Eq. (3) will be investigated in this paper alongside the numerical solution.

3. EXPERIMENTAL CONFIGURATION AND DESCRIPTION

3.1. Dimensional Analysis

The functional relationship for the discharge number in a free-surface vortex is given by

$$\frac{Q}{d^{2.5}\sqrt{g}} = f_2\left(\frac{\Gamma_{\infty}}{wd}, \frac{H}{d}, \beta, W_e, R_R\right) \quad (4)$$

where the dependent term is the discharge number Q_n, Γ_{∞}/wd is the circulation number N_{Γ} , β is a non-dimensional parameter describing the approach flow geometry, W_e is the Weber Number, R_R is the radial Reynolds number, Γ_{∞} is the absolute field circulation, w is the average velocity at the outlet, ρ is the fluid density, σ is the surface tension, g is the gravitational acceleration, b is the inlet width, r_{in} is the distance from the orifice centre to the inlet centroid, H is the approach flow depth and d is the orifice diameter. Regarding hydraulic similitude, the effects of viscosity and surface tension on free-surface vortex flow are generally considered to be negligible if certain predefined hydraulic conditions can be met (Quick 1961). The effect of surface tension (and hence, Weber number) on vortex flow flow can be considered to be negligible in comparison to viscous and gravitational effects as shown by Dagget & Keulegen (1974). Based on the value for Radial Reynolds number, studies have shown that Γ_n, is unaffected when; $R_R > 10^3$ for cylindrical tanks with an orifice (Anwar, 1965) and $R_R > 4 \times 10^4$ for vertically inverted intakes (Anwar & Amphlett, 1980).

3.2. Experimental Setup

Experiments were conducted in a test rig designed to specifically model vortex flow chambers. The test rig (illustrated in Figure 1) consists of a 0.95 m x 0.85 m tank and 0.5 m deep with a 0.1 m circular orifice located centrally resting on a hydraulic bench and storage tank. The upper tank can be adapted with various geometric boundaries to physically simulate vortex chambers. The model geometry analysed in this paper is highlighted in figure 2. The inlet to the test model comprises of a 65 mm bell mouth pipe entrance and 140 mm flow straightening plates which homogenises the incoming velocity profile. Water is circulated through the system by a centrifugal pump (flow range of 0 - 3.5 l/s) and is accurately monitored using a magnetic flow meter (accuracy = $\pm$ 0.1%) with a reading standard deviation of $\pm$ 0.0015 l/s. A depth gauge (graduation error of $\pm$ 0.01 mm) is used to traverse the flow field in order to track the location of the water surface profile. In order to determine the inlet and tangential velocity profiles, particle tracking velocimetry(PTV) is applied on two dimensional horizontal x, y-planes of the vortex by tracking tracer particles (d_p = 0.8 mm) as they intersect a slice of the flow field. This is achieved by illuminating the required plane using a 2mm thick laser light sheet generated by a 532 nm Nd:YAG laser system. Illumination is provided in pulses with frequencies ranging between 10-30 Hz. The light sheet is shaped by passing the 6 mm diameter circular beam profile through appropriate light sheet optics. This optical configuration is arranged in such a way that it is possible to readjust the position and height of the light sheet to probe various sub surface depths z/d of the vortex. The laser pulse duration is extremely short (< 15 ns), which permits the determination of the instantaneous position of the particle in the x, y space when imaged on a CCD camera. The high speed camera (Motion Blitz mini cube) is installed in a compartment which permits perpendicular imaging of the light sheet through a view port on the underside of the tank (see figure 1 for reference).

Particle displacements within each pulse are recorded by acquiring successive images of the particle as it moves through the flow field. The images are calibrated using a predefined geometric grid that is orientated at a particular radius from the orifice and imaged prior to each test. The displacements are manually determined using a cubic spline traced between successive particle images. This tedious but precise method ensures an accurate determination of velocities (± 4%). The displacement vectors determined from the particle images are then used to calculate velocity vectors v_θ and v_r from the time between laser pulses. The main drawback of this PTV configuration is that particle displacements close to the vortex core could not be measured due to an obstruction of the field of view.

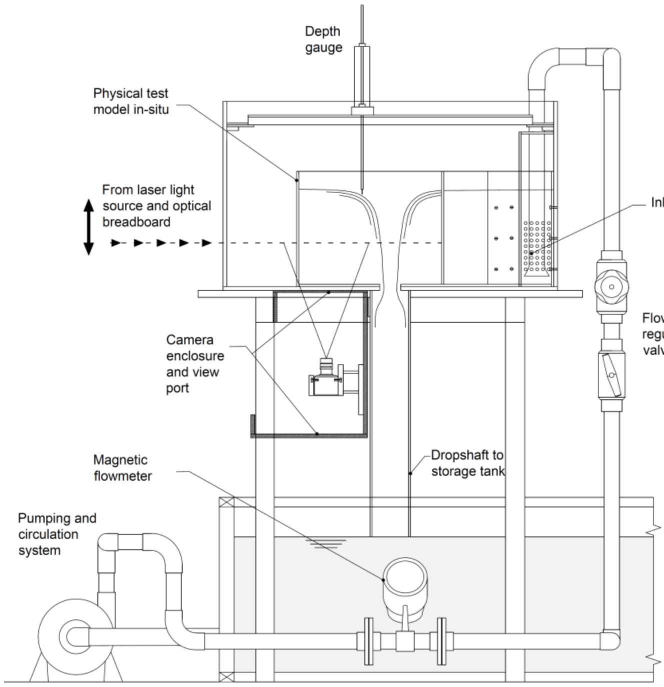

Figure 1 - Illustration of test rig

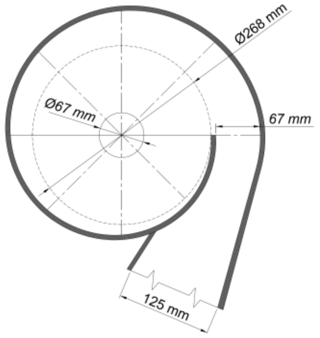

Figure 2 - Model geometry

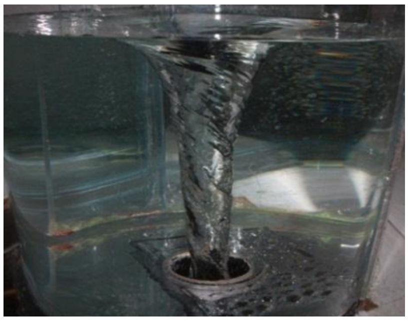

Figure 3 - Vortex air core

3.3. Testing Procedure

In order to carry out an appropriate validation of the numerical model in the vortex chamber, three test physical cases are considered which are based on three equally varying approach flows (H/d and Q) (see table 1). For each approach flow (H/d), the tangential velocity profile is determined using the prescribed PTV technique for various sub surface depths of the vortex (z/d). This data is necessary to validate the numerical model for tangential velocity simulation at a range of sub surface depths. In addition to the velocimetry data, the water surface profile is recorded using the depth gauge apparatus for each approach flow condition.

4. NUMERICAL MODELLING USING ANSYS CFX

4.1. Overview of Multiphase and Turbulence Modelling

The two phase fluid domain is modelled using the Eulerian-Eulerian homogeneous multiphase flow model. In this approach, a single momentum equation is solved for mixture of all fluids and the velocity field is shared among both phases. Individual phase quantities for velocity, density and viscosity are determined using volume fraction α_q. The tracking of the interface is accomplished by solving the volume of fluid (VOF) equation implicitly and the continuity equation iteratively with momentum and pressure. The free-surface is identified for cells containing a volume $\alpha_q = 0.5$. As discussed previously, the flow system in question is highly turbulent containing streamline curvature and therefore particular cases for turbulence modelling must be considered. Standard Reynolds averaged Navier-Stokes (RANS) turbulence models have been commonly found to overestimate the turbulence intensity (Suerich-Gulick et al., 2006) within vortices. Therefore in this analysis, the k-ω based Shear Stress Transport (SST) (Menter, 1994) with and without curvature correction is applied. Curvature correction (CC) (Spalart & Shur, 1997) sensitises the solution in regions of strong curvature by reducing the production of turbulent kinetic energy and increasing its rate of dissipation. Additionally the baseline (BSL) Reynolds stress turbulence models (RSM) is investigated. Despite the additional complexity in solving the individual components of the Reynolds stress tensor and the dissipation rate, this model remains superior in highly straining and anisotropic cases where the eddy-viscosity assumption is no longer valid.

4.2. Modeling Process and Test Cases

The flow in the vortex chamber is simulated using a steady-state, fully separate, multiphase model comprising of water as the primary fluid and air as the secondary fluid. Details of the boundary conditions and test cases are highlighted in Table 1. The solution is independent of the inlet turbulence intensity and is therefore set to a moderate value of 10%.

Table 1 Boundary conditions and test cases

Boundary		Details			
A	**Inlet**	***Boundary type***: Mass flow normal to boundary with hydrostatic pressure defined using CFX expression language (CFL) **Other details:** 10% turbulence intensity			
		Test Cases	*A1*	0.725 l/s	H/d = 1 (67 mm)
			A2	1.677 l/s	H/d = 2 (134 mm)
			A3	3.111 l/s	H/d= 3.5 (235 mm)
B	**Outlet**	**Boundary type**: Opening with entrainment, zero relative pressure			
C	**Walls**	**Boundary type:** No-slip smooth wall boundary			
D	**Opening**	**Boundary type:** Opening with zero pressure and direction			

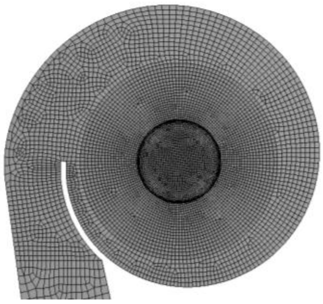

Figure 4 - Plan view of radially structured mesh

Sensitivity analysis was carried out initially on the A3 boundary condition for mesh type (unstructured tetrahedral and structured hexahedral) and density ranging from coarse to fine (minimum cell sizes ranging from 5mm - 1.5mm). Appropriate y+ values are maintained using mesh inflation at vessel walls. Figure 4 is an example of the finely structured hexahedral mesh. A mesh dependency analysis was carried out using (a) the SST model because it is more robust and computationally inexpensive compared to that of the Reynolds stress model. Subsequent sensitivity analysis for turbulence modeling is considered using (b) SST-CC and (c) the baseline Reynolds stress model (RSM). Advection and turbulence is solved using 2nd order upwind schemes. To obtain a converged solution, the target value of the normalised residual for each flow variable was set to 10^{-5}. Initial conditions for the domain are set to zero pressure and velocity (no water or flow) and the false transient method is applied to reach a steady state to best simulate the physical model. For the RSM, the solver is initialised using the results from the SST-CC model.

5. RESULTS AND DISCUSSION

5.1. Initial Observations

The vortices examined in this study (figure 3) appear to be strong in nature with a wide full air core. The approach flow depth was steady in all tests. It is noted that the vortex core becomes transient and oscillated vigorously when the approach flow depth is increased (when N_{Γ} decreases). The Froude numbers ranged from 0.195 - 0.912 for the inlet and 0.077- 1.285 for the outlet. No supercritical approach flow conditions were experienced. The radial Reynolds number varied between 5 x 10^3 - 1.2 x 10^4 suggesting that a full Froude similarity model can be employed when comparing to a prototype. Furthermore, the Weber Number ranged between 90 - 1000 and so the effects of surface tension can be neglected. Another condition specified by Anwar (1965) is that the model should be no smaller than 1/20 for scaling results to the prototype. In this case, the model scale ranged from 1/5 - 1/30 and so the authors are reluctant to assume that scale effects are completely eliminated in this study.

5.2. Solution Sensitivity

Figures 5 and 6 present a summary of the solution sensitivity analysis implemented for the mesh and turbulence model dependence as discussed previously. Figure 5 highlights that the unstructured mesh is detrimental for such an analysis. It is probable that this substantial error is caused by excessive numerical (false) diffusion generated from the poorly aligned grid. From both graphs it can be appreciated that the solution tends to become grid independent when the fine mesh scenario is reached using the SST model. The curvature correction method makes a significant improvement to the SST model, particularly close to the vortex core. The most accurate solution obtained can be observed for the Reynolds stress model. However, discrepancies in the water level at the inlet and vortex core were found to be 22% and 29% respectively. The model also presents a disagreement by over estimating the tangential velocity field. It can be argued that the SST-CC model is competitive with the RSM due to its low computational cost. Both the SST-CC and RSM turbulence models are examined in subsequent sections of this study.

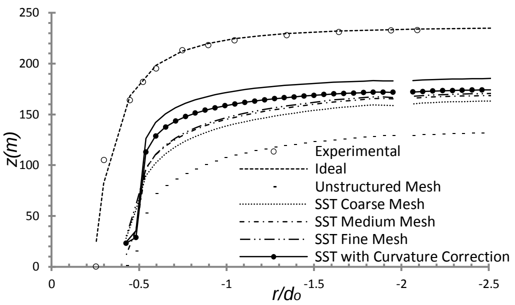

Figure 5 - Sensitivity analysis for the solution of the water surface profile

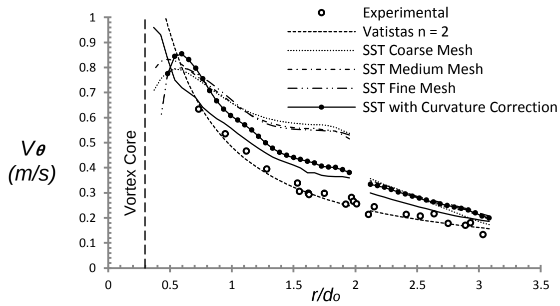

Figure 6 - Sensitivity analysis for the solution of the tangential velocity field

5.3. Water Surface Comparison

Figure 8 presents a comparative analysis for the water surface profile for all test cases with the SST-CC model, RSM and analytical model. The analytical prediction tends to agree well in the far field with discrepancy observable close to the vortex core. Regarding the numerical solution, in all cases the results reveal that the general predicted shape and appearance of the free surface agrees well with the experimental observations. However, the location of the free water surface (RSM results) is under-predicted in each test case (A3 - A1) by approximately 22%, 21% and 26% respectively. Free-surface instabilities are also observable on the iso-surface of the RSM and SST models.

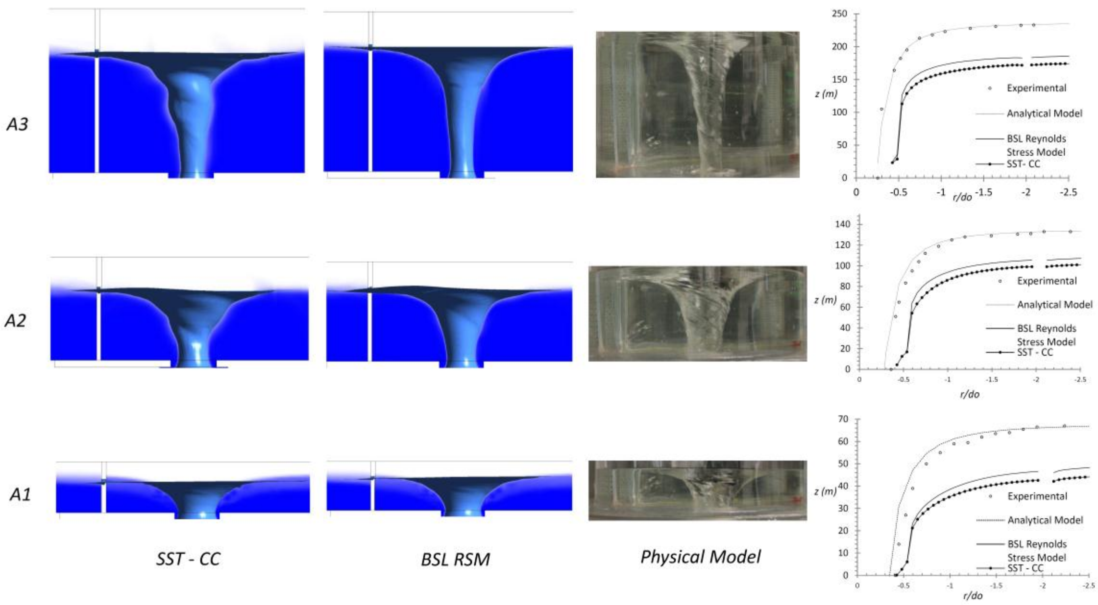

Figure 7 – Summary of comparative analysis for water surfaces between the analytical, SST-CC and RSM physical model for A1, A2 and A3 test cases.

5.4. Tangential Velocity Distribution

A comparison between the experimental model data, Vatistas $n = 2$ analytical model and the numerical simulations of the SST and BSL Reynolds stress models are presented in figures 8, 9, and 10. This comparison is presented for case setting A2 and three sub surface depths. Similar comparisons were found for the A1 and A3 case studies which will not be displayed here.

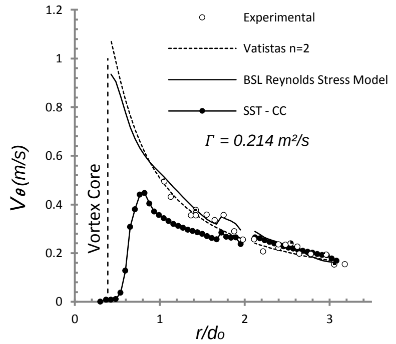

Figure 8 - Tangential velocity profile for $z/H_{in} = 0.26$

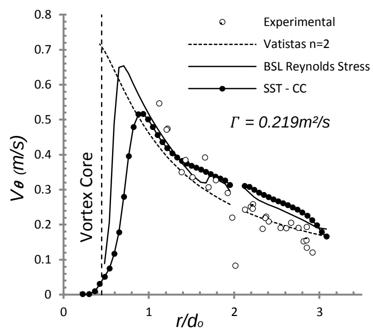

Figure 9 – Tangential velocity profile for $z/H_{in} = 0.56$

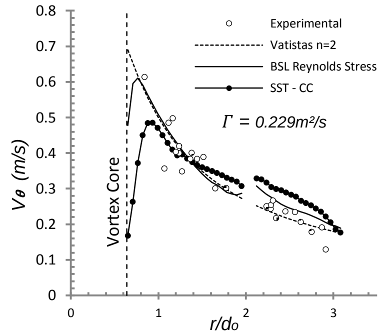

Figure 10 – Tangential velocity profile y $z/H_{in} = 0.75$

The Vatistas n = 2 model agrees well in the far field, however, it is difficult to validate the solution in the near field due to the lack of experimental data at the vortex core. As discussed previously, the Reynolds model outperforms the SST-CC model but suffers high computational expense. Errors in predicting the tangential velocity were found to range from -15% in the far field to + 4% in the near field as a result of a significant under estimation of the air core radius. The SST-CC model suffered errors ranging from -19% in the far field to 8% in the mid-near field. Local transient surface instabilities (observed in figure 7) at the air core interface cannot be fully resolved by the steady state approach resulting in poor convergence. Based on these observations, the authors believe that both the Reynolds stress and the SST-CC models will be substantially improved when a transient simulation is implemented.

6. CONCLUSION

In this work, a numerical-experimental study of the strong free-surface vortex is presented. Three approach flows in a vortex chamber were considered for validation using experimental results from subsurface tangential velocity and water surface profiles. The physical model results indicate that the effects of surface tension and viscosity can be neglected with caution when scaling to the prototype. A sensitivity analysis indicates that an unstructured tetrahedral mesh should be avoided when examining vortex flows due to excessive numerical diffusion. A radially structured mesh centered over the orifice significantly improves the solution. Mesh density gains solution independence when the grid size is approximately 2.5% of the outlet diameter. The turbulence study indicates that the shear stress transport model can result in a poor solution if curvature correction is not accounted for. The Shear stress transport with curvature correction improves the solution by sensitising the model in areas of high streamline curvature and is therefore relatively inexpensive. The Reynolds stress model outperforms the SST-CC model slightly with the cost of higher computational effort. Both the SST-CC and the RSM suffered errors in the region of 22-29% in predicting the air water interface and 15 – 19% in predicting the tangential velocity field. It is concluded that free-surface instabilities and oscillations at the air core interface are preventing the steady state model from approaching a converged solution and therefore transient analysis is required. The results provided by the Vatistas $n = 2$ analytical model exhibit good agreement with the water surface and tangential velocity in the far field but diverge slightly at the vortex core.

7. ACKNOWLEDGEMENTS

The authors would like to express their gratitude to the Irish Research Council for the financial support of this work.

8. REFERENCES

Anwar, H. O. (1965), *Flow in a Free Vortex, Water Power*, 17(4), 153-161.

Anwar, H.O., Amphlett, M.B. (1980), *Vortices at Vertically Inverted Intake*, IAHR, Journal of Hydraulic Research, 1980(2).

Bayeul-Lainé A.C., Simonet. S., Bois, G., and Issa, A. (2012), *Two-phase Numerical Study of the Flow Field Formed in Water Pump Sump: Influence of Air Entrainment*, 26th IAHR Symposium on Hydraulic Machinery and Systems, 2012 Beijing, China.

Binnie, A.M., and Davidson, J.F. (1949), *The Flow Under Gravity of a Swirling Fluid*, Proc. Roy. Soc. A, 199.

Cristofano, L., Nobili, M. and Caruso, G. (2009), *Numerical Evaluation of Gas Core Length in Free Surface Vortices,* 32nd UIT (Italian Union of Thermo-fluid-dynamics) Heat Transfer Conference, IOP Publising, Journal of Physics: Conferences Series, 547.

Brocard, D.N., Beauchamp, C.H. and Hecker G.E. (1982), *Analytic Predictions of Circulation and Vortices at Intakes*, Electric Power research Institute, Research Project 1199-8.

Dagget, L.L., Keulegen, G.H. (1974), *Similitude in Free-Surface Vortex Formations*, ASCE. J. Hydraul. Div., HY11. 1565-1581.

Drioli, C. (1947), *Su un particolare tipo di imbocco per pozzi di scarico (scaricatore idraulico a vortice)*, L'Energia Elettrica, 24(10), 447–452 (in Italian).

Lei, G., Jian-min, Z., Jian-Gang, C., Fei, Y., Lu., Lie. *(2011), 3-D numerical simulation research of flow in a vortex drop shaft which have two volute chambers with aeration.* Proceedings of the 34th IAHR World Congress, Brisbane Australia, 2011, 1779-1786.

Hite, J. E. and Mih, W. C. (1994), *Velocity of air-core vortices at hydraulic intakes*, Journal of Hydraulic Engineering, 120 (3), 284-297.

Hai-feng, L., Hong-xun, C., and Zheng, Ma. (2008), *Experimental and numerical investigation of free surface vortex*, Journal of Hydrodynamics, 20(4), 485.

Menter, F. R., (1994), *Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications*, AIAA Journal, 32(8), 1598-1605.

Okamura, T., Kamemoto, K., & Matsui, J. (2007), *CFD prediction and model experiment on suction vortices in pump sump*, Proceedings of the 9th Asian International Conference on Fluid Machinery, October 16- October 19, 2007, Jeju, Korea.

Quick, M.C. (1961), *A Study of the Free Spiral Vortex*, PhD Thesis University of Bristol.

Rankine (1858), *W.J.M. Manual of Applied Mechanics*, C. Griffen Co., London.

Rajendran, V.P., Constantinescu, S.G., Patel, V.C. (1999), *Experimental Validation of Numerical pump intake-bays*, Journal of Hydraulic Engineering, 125(9), 1119-1125.

Skerlavaj, A., Skerget, L., Ravnik, J. and Lipej, A. (2013). *Predicting Free-surface Vortices with Single –Phase Simulations*, Engineering Applications of Computational Fluid Dynamics, 8(2), 193-210.

Spalart, P.R., and Shur. M. L. (1997), *On the Sensitization of Turbulence Models to Rotation and Curvature,* Aerospace Science and Technology, 1 (5), 297–302.

Suerich-Gulick, F., Gaskin, S. *Experimental and Numerical Analysis of Free Surface Vortices at a Hydropower Intake,* Proceedings of the 7th International Conference on Hydroscience and Engineering, ICHE 2006.

Suerich-Gulick, F., Gaskin, S., Villeneuve, M., Parkinson, E. (2007), *Numerical Simulation of a Free Surface Vortex Formed in the Wake of a Pier at a Hydropower Intake*. 32nd Congress of IAHR, July 1-6 2007, Venice, Italy, pp.1-10.

Tokyay, T. and Constantinescu, S.G. (2005), *LES and RANS simulations of flow in a realistic pump-intake*, World Water and Environmental Resources Congress, May 15-19, Alaska.

Trivellato, F. (1995), *On the Near Field Representation in a Free Surface Vortex*, 12th Australasian Fluid Mechanics Conference, The University of Sydney, Australia.

Tanweer, S.D., Gahlot, V.K. (2010) *Simulation of Flow through a Pump and its Validation*, IJRRAS, 4(1)

Vatistas, G.H., Kozel, V., and Mih, W.C. (1991), *A Simpler Model for Concentrated Vortices,* Experiments in Fluids, Vol. 11, 73-76.

Vatistas, G.H., Lin, S., and Kwok, C.K. (1986), *Theoretical and Experimental Studies on Vortex Chamber Flows*, AIAA Journal, 24(4), 635-642.

Wang, Y. K., Jiang, C. B., and Liang, D. F. (2011), *Comparison between empirical formulae of intake vortices*, Journal of Hydraulic Research, 49 (1), 113-116.