Research Article

Numerical and Experimental Study of Pump Sump Flows

Wei-Liang Chuang,¹ Shih-Chun Hsiao,¹ and Kao-Shu Hwang²

¹ Department of Hydraulic and Ocean Engineering, National Cheng Kung University, Tainan 701, Taiwan

² Tainan Hydraulics Laboratory, National Cheng Kung University, Tainan 701, Taiwan

Correspondence should be addressed to Shih-Chun Hsiao; schsiao@mail.ncku.edu.tw

Received 21 August 2013; Revised 24 January 2014; Accepted 7 February 2014; Published 24 April 2014

Academic Editor: Jian Guo Zhou

Copyright © 2014 Wei-Liang Chuang et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

The present study analyzes pump sump flows with various discharges and gate submergence. Investigations using a three-dimensional large eddy simulation model and an acoustic Doppler velocimeter are performed. Flow patterns and velocity profiles in the approaching flow are shown to describe the flow features caused by various discharges and gate submergence. The variation of a large-scale spanwise vortex behind a sluice gate is examined and discussed. The suction effect on approaching flow near the pipe column is examined using numerical modeling. To gain more understanding of the vortices variation, a comparison between time-averaged and instantaneous flow patterns is numerically conducted. Additionally, swirl angle, a widely used index for evaluating pump efficiency, is experimentally and numerically examined under various flow conditions. The results indicate that the pump becomes less efficient with increasing discharge and gate submergence. The fluctuation of the free surface over the pump sump is also discussed.

1. Introduction

Large-scale pumps with capacities of 1–10 m³/s are extensively used in circulating cooling system for electric power plants, flood-control systems, and sewage-treatment plants. A typical pump sump consists of a suction pipe and a sump. Designers may consider various structures or components, such as a sluice gate or screening, for specific applications. Multipump systems are widely used, but few studies have addressed this configuration due to the complexity and high computation cost [1–3]. Ansar and Nakato [4] and Chuang and Hsiao [5] found that cross flow induced by large circulating flow in a pump sump may seriously degrade pump efficiency. Poor pump sump and intake structure designs thus often lead to serious problems [6]. There are some guidelines for engineering applications, including the “*The Hydraulic Design of Pump Sumps and Intakes*” published by the British Hydromechanics Research Institute [7] and the “*American National Standard for Pump Intake Design*” [8]. Flow problems, such as vortices induced by a fluid passing through obstructions, can remarkably reduce pump

efficiency or even damage pump components [9, 10]. The cavitation induced by an air-entraining vortex may seriously damage the impeller fans. Nonuniform flow in the suction pipe may oscillate the impeller or other mechanical components, increasing maintenance costs [10]. Some common solutions used for eliminating undesirable flow conditions include flow-straightening devices such as baffle bars and perforated plates. Local solutions for suppressing the formation of subsurface vortices include the installation of horizontal floor-corner fillets, vertical sidewall-backwall corner fillets, and floor- and backwall-attached splitters [11, 12].

Many guidelines have been updated based on recent research effort. However, these guidelines are only applicable to sumps with a simple geometry. Most pump sump design problems are site specific. Moreover, with the advanced progress achieved on the accessories in the approaching flow, the pump intake problem is difficult from the viewpoint of flow characteristics. For instance, to improve filtering efficiency, travelling water screens have gradually been changed from a traditional through-flow screen type to a center-flow screen type, which can degrade flow conditions

[10]. Ansar and Nakato [4] studied three-dimensional (3D) pump intake flows with and without a cross flow. In their work, acoustic Doppler velocimetry (ADV) was employed to examine the flow pattern in the approaching flow. Hwang and Yang [10] experimentally evaluated pump efficiency with various flow-straightening devices beneath the bell mouth. Their measurements using ADV and a swirl meter provide valuable information for practical applications. With regard to numerical modeling, Constantinescu and Patel [13] developed a 3D multiblock Reynolds-averaged Navier-Stokes (RANS) model with the choice of a near-wall two-equation turbulence model ($k-\varepsilon$ and $k-\omega$) for high and low Reynolds numbers. Instead of marking progress to viscous model, Ansar and Nakato [4] adopted an inviscid solution to establish the flow patterns in the approaching flow with and without cross flow. The simulated patterns and quantities agreed with the measurements, but not for the region close to the solid boundary. Thus, they suggested that the viscosity effect be considered when numerically modeling a pump sump flow. Li et al. [14] applied a RANS model to compare the streamwise velocity distribution with measurements not only in the approaching channel, but also above the pump throat. They found that the steady-state solution fails to describe the flows at the pump throat in the cross-flow case. They conjectured that the nature of the RANS model may limit its usefulness for investigating pump sump flows. Tokyay and Constantinescu [15] compared the predictions of large eddy simulation (LES) and RANS models with measurements taken using particle image velocimetry (PIV) [16]. They showed that a nondissipative LES model produces better predictions of the statistical velocity magnitude and vorticity, streamlines, and turbulent kinetic energy distribution than those obtained using the shear-stress transport (SST) RANS model. Chuang and Hsiao [5] numerically studied the same issue [4] by including the effect of fluid viscosity and more realistic simulation conditions. Their results showed that viscosity affects the prediction of flow patterns and that the streamwise velocity can be better captured by including cross flow (i.e., considering a larger domain) instead of imposing a given boundary condition.

The objective of the present study is to experimentally and numerically investigate the influence of the submergence sluice gate and discharge on pump sump flows. Measurements in an approaching flow are conducted using ADV in a scaled-down circulating-water pump sump model. Time-averaged flow patterns and streamwise and vertical velocity profiles are shown. To evaluate pump efficiency, the most widely used apparatus, swirl meter, is applied, and its motion is recorded by a camera. The time-averaged and maximum swirl angles under various flow conditions are calculated and a comparison is made with numerical results. For better understanding of the flow field where measurements are not feasible, a 3D numerical model that considers a realistic suction bell is also performed. The simulation results are first validated with measurements through velocity profiles and compared with measured flow patterns. The flow field, including the vorticity inside the suction pipe, is investigated at high spatial resolution. The free surface fluctuation in the pump sump is also examined.

2. Intake Model Description

A 1:10-scale intake model of a circulating-water pump sump similar to that utilized in Shen-Ho Power Plant, Keelung, Taiwan, was built. Since Shen-Ho Power Plant was built along a rocky coastline, the opening of the sluice gate is thus site-restricted, with an opening height of 3.63 m. This small opening size may disturb the uniformity of approaching flow, lowering pump efficiency. One objective of this paper is to better understand how the submergence of the sluice gate affects the pump efficiency. Of note, the scale effect is of great concern when conducting hydraulic model tests. Padmanabhan and Hecker [17] found no significant scale effects on the formation of vortices when the approaching Reynolds number, based on submergence and approaching flow velocity, is greater than 3×10^4 . Jain et al. [18] studied vortex formation at vertical pipe intakes. They found that there is no influence of surface tension on critical submergence when the Weber number, based on diameter and pipe-intake velocity, is greater than 120. In this paper, all studied cases meet the above criteria to minimize possible scale effects.

For sump flow problems, two notable air-entrained problems are free surface vortex and subsurface (wall-induced) vortex. As reported by Quick [9], the formation of air-entraining vortex requires two conditions.

- (1) Vorticity exists in the approaching flow.
- (2) The submergence Froude number, based on submergence and pipe-intake velocity, must be greater than 1.

Most layouts of pump sump may easily meet the first requirement. But for the second one, a supercritical approaching flow must be excluded in the design stage. In operation, care should be taken when water depth is lower than optimal elevation. In this study, as listed in Table 1, the submergence Froude numbers are all less than 1. Among them, the maximum value 0.79 is the only numerically investigated case, of which the flowrate is twice the operating flowrate. Also, none of the air-entrained vortices was ever reported throughout the experiments of case study. Therefore, there is no need to model multiphase flow and air entrainment in simulation.

The measurements were carried out in the Tainan Hydraulics Laboratory (THL), National Cheng Kung University, Tainan, Taiwan. As shown in Figures 1(a) and 1(b), the hydraulic model mainly consists of a tank, a forebay, a sluice gate, a suction pipe (a cylindrical pipe connected with a suction bell mouth), and a pipe column. The diameter of cylindrical pipe was $D_p = 160$ mm and the width of the approaching channel was $W = 427$ mm. The mean water depth (H) was kept at 450 mm throughout the experiments. The intake submergence (h_s) was 350 mm.

The design discharge, $Q = 2.5 \times 10^{-2} \text{ m}^3/\text{s}$, is the optimal operating discharge. The maximum-limited pumping discharge is $Q = 3.7 \times 10^{-2} \text{ m}^3/\text{s}$. A higher discharge of $Q = 5.1 \times 10^{-2} \text{ m}^3/\text{s}$ was used only in the numerical simulation to investigate the effect of the Reynolds number. Since the Shen-Ho Power Plant is located along a rocky coastline,

TABLE 1: Summary of case study.

Case	S (mm)	Q (m ³ /s)	$F_h \frac{V_b}{\sqrt{gh_s}}$	$R_a \frac{V_a h_s}{\nu}$	ADV measurement	Swirl angle measurement	Numerical simulation
SN	0	2.5×10^{-2}	0.38	4.9×10^4		✓	
SL	87	2.5×10^{-2}	0.38	4.9×10^4	✓	✓	✓
SM	87	3.1×10^{-2}	0.47	6.1×10^4	✓	✓	✓
SH	87	3.7×10^{-2}	0.56	7.3×10^4	✓	✓	✓
SE	87	5.1×10^{-2}	0.79	1.0×10^5			✓
SLA	129	2.5×10^{-2}	0.38	4.9×10^4	✓	✓	✓
SMA	129	3.1×10^{-2}	0.47	6.1×10^4		✓	
SHA	129	3.7×10^{-2}	0.56	7.3×10^4		✓	
SLB	171	2.5×10^{-2}	0.38	4.9×10^4	✓	✓	✓
SMB	171	3.1×10^{-2}	0.47	6.1×10^4		✓	
SHB	171	3.7×10^{-2}	0.56	7.3×10^4		✓	

the opening of the sluice gate is restricted to a height of 3.63 m, and thus 87 mm (in the scaled-down model) is the minimum submergence of the sluice gate. Details of the case studies are summarized in Table 1. Note that in Table 1, F_h is the submergence Froude number based on the intake submergence and the mean axial velocities at the bell mouth (V_b); R_a is the approaching flow Reynolds number related to the intake submergence (h_s) and the time-averaged velocity at the approach channel (V_a). ν is the kinematic viscosity.

3. Acoustic Doppler Velocimetry

ADV measures instantaneous 3D velocities based on the acoustic Doppler principle. The velocities are derived from the frequency difference caused by scattering waves as acoustic waves bounce off floating particle tracers in the water. Typically, an ADV sensor mainly consists of three components: a measurement probe, a signal-conditioning module, and a signal-processing module. The probe has one transmitting and three receiving transducers. The azimuth of the interval between each receiving transducer is 120°. The angle between each receiver, transmitter, and sampling volume is 30°. The measuring volume is located 50 mm below the probe.

The ADV sensor, manufactured by SonTek (ADV laboratory 16 MHz), has a 25 Hz sampling rate in a sampling volume of about 90 mm³. The velocity measurement accuracy is around ± 3 mm/s. The density of seeding particles is crucial for the quality of the measurement signal. In the measurement process, the signal-to-noise ratio (SNR) should be controlled to be above the suggested value of 15 dB. The method reported by Rusello et al. [19] was further used to control the system noise levels. The data correlation was 95%, higher than the 85% suggested by the manufacturer. To ensure the accuracy in measuring the turbulent flow, a so-called spike [20, 21] in the record should be eliminated. The method developed by Goring and Nikora [21] was adopted to detect and replace spikes in ADV data sequences.

Two ADV sensors (see Figure 1(b)) were utilized throughout the experimental procedure. One ADV sensor was installed on a 3D movable trolley. A fine-tuning trolley was

regulated by a screw rod, with pitches of 6.5 mm and 1.3 mm per rotation in the y -axis and z -axis directions, respectively. A total of 395 measurement points were distributed between the sluice gate and the front face of the suction-pipe-supporting deck in the approaching channel, as shown in Figure 2. Note that sections X13 to X16 were selected to examine the velocity profile near the suction pipe by numerical modeling. Based on a previous study [22], 120 s was chosen as the sampling time for each measurement point. However, doubt must be given over such time period due to different layout and flow conditions. To address this issue, another ADV sensor, located 100 mm above the floor at the upstream centerline of the channel, served as a reference probe. The time-series velocities and their standard deviations were compared among repeated measurements. The results (not shown here) verified that the steady-state data collected within 120 s are confident.

4. Swirl Meter and Camera

Some studies [16, 23, 24] have employed PIV to quantify the flow near the suction pipe and entrance. They examined the flow fields near the suction bell, where the flow pattern is supposed to much more directly reflect the influence of the approaching flow. However, the flow inside the suction pipe remains unclear due to the difficulty of implementing a nonintrusive measurement technique. The pump efficiency here is well determined by flow conditions, the most crucial index in design or redesign. Practical applications utilize the swirl meter as an indicator [10, 22]. In the present study, a swirl meter with low-friction bearing was installed inside the suction pipe. The swirl meter had four rectangular fans, one of which was painted red as a reference. The definite location, elevation, and diameter of swirl meter referred to in [8] are shown in Figure 3. Generally, a swirl meter often randomly revolves during pump operation. In this study, the revolving motion was recorded by a camera (SONY-NV500) capable of capturing 30 frames per second. The observation period was 10 min. In the process, several time points were tracked when the reference fan passed through a checkpoint. Figure 4 shows a sample record. According to the time-series record,

(a)

(b)

FIGURE 1: (a) Schematic diagram of intake model and accessory apparatuses. (b) Photograph of 1:10-scale intake model of circulating-water pump similar to that in Shen-Ho Power Plant. Implementations of ADV sensors, fine-tuning trolley, and swirl meter (inside pipe column) are shown.

the instantaneous and time-averaged revolutions per second (rps) can be determined.

The swirl angle is an important and widely used index for empirically practical evaluating pump efficiency. The nonuniformity or unsteadiness of the approaching flow contributes to the swirl angle of the flow in the suction pipe. The swirl angle, Φ , is defined as

$$\Phi = \tan^{-1} \left(\frac{V_{\Phi}}{V_P} \right) = \tan^{-1} \left(\frac{\pi N D_P}{V_P} \right), \quad (1)$$

where V_{Φ} is the tangential velocity, V_P is the mean axial velocity ($= 4Q/\pi D_P^2$) of the cylindrical pipe flow, and N is the rps of the reference fan of the swirl meter.

5. Flowmeter

An electromagnetic flowmeter (Danfoss MAG 3100 W) was employed at the end of the pipe line. To ensure that discharge can be precisely and stably controlled in the experimental process, calibration was done using a 1.2 m^3 standard steel tank. A series of tests verified the discharge control, with an error of around $\pm 0.5\%$ during pump operation.

FIGURE 2: Locations of measurement points: (a) side view and (b) top view. Note that X1 is at $x = 60$ mm and Z1 is at $z = 20$ mm. The reference point is 100 mm above the bottom at the centerline of the pump sump (i.e., Y3). All units are in mm.

FIGURE 3: Diagram of swirl meter implementation, diameter, and location. More details can be found in [8].

6. Numerical Model

A 3D viscous numerical model developed by Los Alamos National Laboratory (LANL) in the United States, TRUCHAS, was applied to simulate the pump sump flows. TRUCHAS provides a multiphysics computational platform for modeling a wide range of physical problems involving materials and/or incompressible flow. In recent years, the nature of 3D solver made TRUCHAS more popular to those scholars who are interested in hydrodynamics. Modules, such as interface tracking, moving boundary, turbulence model, and wavemaker, were modified, refined, or implemented to gain wider range of applications or develop various divisions. Several studies have validated TRUCHAS simulations using measurements and used the simulations to gain insight into wave-structure interactions [25–28], multiphase flows [28, 29], and moving boundary [30, 31].

FIGURE 4: Sample of time-series record of swirl meter. The label +1 on vertical coordinate marks the counterclockwise $\pi/2$ revolution and -1 marks the clockwise $\pi/2$ revolution.

7. Governing Equations and LES Model

TRUCHAS solves Navier-Stokes equations with LES turbulence modeling. The main idea of LES is to explicitly solve the large-scale (grid-scale) eddy motion by filtering out the small-scale eddy motion and modeling the small-scale turbulent fluctuations. Here, the Navier-Stokes equations were filtered using the method proposed by Deardorff [32], which spatially filters out the variables with a filter function G with filtered variable $\bar{\phi}$ defined as

$$\bar{\phi}(x) = \int \phi(x') G(x, x') dx', \quad (2)$$

where G is defined as

$$G(x, x') = \begin{cases} \frac{1}{V} & \text{for } x' \in V \\ 0 & \text{otherwise,} \end{cases} \quad (3)$$

where $\forall$ is the volume of a computational cell. By applying (2) and (3) to Navier-Stokes equations, the filtered mass and momentum equations are, respectively, derived as

$$\begin{aligned} \frac{\partial \bar{u}_i}{\partial x_i} &= 0, \\ \frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} \\ &= -\frac{\partial \bar{P}}{\partial x_i} + \frac{\partial (\bar{\tau}_{ij})_l}{\partial x_j} + \frac{\partial (\bar{\tau}_{ij})_t}{\partial x_j} + \rho g_i, \end{aligned} \quad (4)$$

where subscripts $i, j = 1, 2, 3$ refer to the directions of 3D flows. $\bar{u}_i$ indicates the i th component of the filtered velocity field, $\bar{P}$ is the filtered pressure, ρ is the density, and g_i is the i th component of the gravitational field. $(\bar{\tau}_{ij})_l$ is the filtered molecular viscous stress tensor. $(\bar{\tau}_{ij})_t$ is the subgrid scale Reynolds stress tensor that contains the local average of the small-scale field to be modeled. A widely used subgrid scale model (SGS model) is the Smagorinsky model [33]:

$$(\bar{\tau}_{ij})_t = -\rho \bar{u}'_i \bar{u}'_j + \frac{1}{3} \rho \delta_{ij} \bar{u}'_k \bar{u}'_k = \rho \nu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right), \quad (5)$$

where δ_{ij} is the Kronecker delta operator, $\bar{u}'_i \bar{u}'_j$ is the turbulence shear-stress component, and ν_t is the sub-grid-scale eddy viscosity, which is modeled as

$$\nu_t = \ell_s^2 \bar{S} = (C_s \Delta)^2 (2 \bar{S}_{ij} \bar{S}_{ij})^{0.5}, \quad (6)$$

where $\bar{S}_{ij}$ is the filtered rate of strain, $\bar{S}_{ij} = 0.5(\partial \bar{u}_i / \partial x_j + \partial \bar{u}_j / \partial x_i)$, and ℓ_s is the Smagorinsky length scale defined as the product of the Smagorinsky coefficient C_s and the filter width Δ . C_s is a model parameter whose value ranges from 0.1 to 0.2 depending on the types of flow problem. Under the isotropic turbulence condition ($C_s \sim 0.2$), tests have confirmed that on $O(5)$ Reynolds number with no obvious difference was found due to different values of C_s in above suggested range. Therefore, C_s is set to 0.15 in this study. To well resolve the near-wall viscous sublayer region, a near-wall damping function, derived by Cabot and Moin [34], is applied to approximate the eddy viscosity at the first cell in the vicinity of the wall. The near-wall treatment is expressed as

$$\begin{aligned} \frac{\nu_w}{\nu_t} &= \kappa y_w^+ \left(1 - e^{-y_w^+/A} \right)^2, \\ y_w^+ &= \frac{y_w u_\tau}{\nu}, \end{aligned} \quad (7)$$

where ν_w is the near-wall eddy viscosity, ν is the kinematic viscosity, u_τ is the shear velocity, κ is the von Karman's constant ($\kappa = 0.41$), $A = 19$, and y_w is the distance from the solid wall to the center of the first cell adjacent to the wall.

8. Finite Volume Method

The finite volume method [35, 36] is applied to discretize the governing equations in the computational domain consisting

of hybrid structured and unstructured meshes. An arbitrary quantity ϕ in the general conservation equation is defined as

$$\frac{\partial \phi}{\partial t} + \nabla \cdot (\mathbf{u} \phi) = S(\phi), \quad (8)$$

where $\mathbf{u}$ is the velocity vector and S is the source term related to ϕ . Integrating (8) over any volume with the aid of Gauss divergence theorem to transform the volume integral into a surface integral yields

$$\int \frac{\partial \phi}{\partial t} dV + \oint (\mathbf{u} \phi) dA = \int S(\phi) dV. \quad (9)$$

Then, (9) can be discretized as

$$\frac{\phi_i^{n+1} - \phi_i^n}{\Delta t} + \frac{1}{\forall_i} \sum_F [\mathbf{A} \cdot \mathbf{u}^n]_F [\phi]_F^n = S_i^n, \quad (10)$$

where n denotes the n th time step, $\forall_i$ is the i th cell volume, subscript F is the cell surface, $\mathbf{A}$ is the surface area vector, and Δt ($t_i^{n+1} - t_i^n$) is the time step increment. ϕ represents the average arbitrary quantity in the i th cell volume. On the left-hand side of (10), the surface integral is approximated as a summation over the cell faces. ϕ_i and S_i can be, respectively, expressed as

$$\begin{aligned} \phi_i &= \frac{\int \phi d\forall}{\int d\forall} = \frac{1}{\forall_i} \int \phi d\forall, \\ S_i &= \frac{\int S d\forall}{\int d\forall} = \frac{1}{\forall_i} \int S d\forall. \end{aligned} \quad (11)$$

The integral form of momentum equation can be readily obtained as

$$\begin{aligned} \frac{\phi_i^{n+1} - \phi_i^n}{\Delta t} + \int_A (\rho \mathbf{u} \phi) dA \\ = \int_A (\mu \nabla \phi) dA + \int_{C \cdot \forall} S_\phi d\forall, \end{aligned} \quad (12)$$

where ϕ_i is the velocity specified in the momentum equation and S_ϕ is the gravitational acceleration in the z direction.

9. Two-Step Projection Method

The momentum equation is solved using the two-step projection method proposed by Chorin [37, 38]:

$$\frac{\rho^{n+1} \mathbf{u}^* - \rho^n \mathbf{u}^n}{\Delta t} = -\nabla \cdot (\rho \mathbf{u} \mathbf{u})^n + \nabla \cdot (\mu^n (\nabla \mathbf{u} + \nabla^T \mathbf{u})^n), \quad (13)$$

$$\frac{\rho^{n+1} \mathbf{u}^{n+1} - \rho^{n+1} \mathbf{u}^*}{\Delta t} = -\nabla P^{n+1} + \rho^{n+1} \mathbf{g}. \quad (14)$$

Equation (13) is called the predictor step and (14) is called the corrector step. Interim velocity $\mathbf{u}^*$ can be solved explicitly

using (13), with ρ^{n+1} derived from volume of fluid (VOF) equation. Equation (14) relates $\mathbf{u}^{n+1}$ to $\mathbf{u}^*$. Combining (13) with (14) yields

$$\nabla \cdot \frac{\nabla P^{n+1}}{\rho^{n+1}} = \nabla \cdot \left(\frac{\mathbf{u}^*}{\Delta t} + \mathbf{g} \right). \quad (15)$$

Equation (15) is the Poisson pressure equation. New time step pressure P^{n+1} can be solved from (15), and cell-face pressure gradient ∇P_F^{n+1} is calculated from a stencil corresponding to that of the density interpolation to faces. Equation (14) is then used to calculate a solenoidal face velocity field:

$$\mathbf{u}_F^{n+1} = \mathbf{u}_F^* - \Delta t \left(\frac{\nabla P_F^{n+1}}{\rho_F^{n+1}} - \mathbf{g}_F \right). \quad (16)$$

Since we use forward differencing in time and upwind scheme in space, the accuracy of this algorithm is first order in time and space. From the numerical experiments and several numerical simulations of breaking waves [25, 26, 28], this accuracy is high enough for studying the complex hydrodynamics problems.

10. Volume of Fluid

Unlike previous numerical studies [1, 4, 13, 16] in which the free surface was treated as a flat plane, TRUCHAS is here implemented with VOF to simulate the free surface. The method adopted in this study was originally developed by Hirt and Nichols [39]. Chuang and Hsiao [5] showed that the free surface is strongly disturbed under the conditions of a high Reynolds number or low water depth, especially with the latter. In modeling, fluids are assumed to be immiscible. The density of the m th fluid is ρ_m . Expressing ρ_m by volume fraction f_m based on the mass conservation equation, the VOF equation can be written as

$$\frac{\partial f_m}{\partial t} + \nabla \cdot (f_m \mathbf{u}) = 0. \quad (17)$$

Equation (17) can be discretized based on (10) yielding

$$f_m^{n+1} = f_m^n - \frac{1}{\nabla} \sum_f \delta t [\mathbf{A} \cdot \mathbf{u}^n]_f [f_m]_f^n. \quad (18)$$

The volume fraction f_m is bounded by $0 \leq f_m \leq 1$. In this study, the background is assumed to be void. Therefore, f_m directly represents the ratio between water and the void fraction in the local cell. $f_m = 1$ denotes that local cell is full of water. This study defines $f_m = 0.5$ as the wave-void interface, that is, the free surface.

Once f_m is determined, piecewise linear interface calculation (PLIC) will establish further step to reconstruct the free surface. The fluid-fluid interface within a cell is assumed to be planar. For more details refer to Wu [25] and Hu et al. [27].

11. Boundary Conditions

In this study, the no-slip condition was imposed on the solid wall boundary. The inflow and outflow boundary conditions

were velocity Dirichlet boundary conditions, which specify a determined value on a specific surface and give rise in flow-in or flow-out boundary, that is, uniform flow condition given at the approaching flow inlet and intake-pipe outlet. When solving VOF equation (18) for a specified surface, $\mathbf{u}^n$ and $\mathbf{u}_f^n$ are given determined values, as well as the predicted velocity $\mathbf{u}^*$. That is, no other boundary conditions are required.

To obtain the stable solution, the standard von Neumann stability analysis states that C_r , defined as $\text{Max}(|\mathbf{u}|)(\delta t/d\zeta)$, where δt is the time step restricted by the advection term and $d\zeta$ is a measure of cell size, should be between 0.5 and 0.85. To maximize the time step, a value of 0.85 was chosen.

12. Computational Meshes

Cubit software (Sandia National Laboratories) was adopted to generate computational meshes. Cubit is capable of generating structured, unstructured, or hybrid meshes with a complex geometry. In this study, the geometry included a complex bell-shaped intake that was effectively resolved using both structured and unstructured meshes. As shown in Figure 5, a fine mesh was used near the solid wall region to capture the boundary-layer feature and small-scale fluid physics. Other regions were filled with a coarser mesh. The size ratio between two adjacent grids was ensured to be no smaller than 1/7.2, compared to the suggested ratio of 1/10.

13. Mesh Independence and Convergence

Mesh independence was examined by comparing the results from mesh systems with 2 million, 1.2 million, and 0.5 million grid cells. The influence of grid size on the velocity profiles was analyzed in the approaching flow for case SL. This case was chosen because the detailed experimental measurements were also available. The computation results indicated that the difference among the mesh systems in terms of velocity profiles in the approaching flow was less than 1%. Considering moderate resolution and efficiency, the mesh system with 1.2 million grids was adopted in this study.

Convergence is of great concern for the LES approach. Wu [25] suggested that at least a 10×10 grid is required to resolve the local largest eddy. However, ADV measurement provided poor resolution, so the size of the local largest eddy was unknown. This issue was addressed by Wu et al. [28], who also applied LES model to investigate wave-structure interaction through macroscopic and microscopic modeling. Gullbrand and Chow [40] gave two suggestions for selecting the appropriate grid size in LES modeling. First, the grid must be able to resolve the important physical characteristics of the flow field. Second, the grid should be fine enough to avoid a solution that is remarkably affected by numerical errors. For the mesh systems used in this study, the resolution tends to converge, and the accuracy and quality of the numerical results are good compared to measurements in terms of the pump sump flows for various gate submergence and Reynolds numbers.

FIGURE 5: (a) Overview of computational mesh. (b) Detailed mesh pattern on section parallel to backwall through centerline of suction pipe.

14. Computational Resources

Due to the huge computational burden for a 3D problem, the computation was distributed to 24 processors over a cluster-based platform. The execution time ranged from 10 days (case SL) to 18 days (case SE), mainly depending on the Reynolds number. The time-averaged stable solution was determined from the variation of time-series velocities. In this study, the acceptable variable magnitude was less than 0.1%.

15. Time-Averaged and Instantaneous Flow Patterns in Approaching Flow

Figures 6, 7, and 8 compare the time-averaged flow patterns between numerical and experimental results on the x - z planes in sections Y1, Y3, and Y5, respectively. To clearly show the flow patterns from numerical results, only the streamline is shown, with the data near the pipe removed. Note that the horizontal line at $z = 450$ mm marks the mean water depth.

Flow patterns in sections Y1, Y3, and Y5 have three main flow-disturbing geometries: sluice gate, bottom step, and pump suction. Taking Figure 6 for example, the flow separates at the tip of the sluice gate. The return flow occurs near the free surface on the lee side of the gate and a large counterclockwise vortex forms. The bottom step forces the near-bottom water body to move upward. For the water body approaching the pipe, the pattern of streamlines, converging toward the entrance caused by suction, becomes much more visible. For the flow field near the bell mouth, the vertical velocity component increases in the negative z -axis direction, and the horizontal velocity component increases in the streamwise (defined as positive x -axis) direction in the lower water body.

In Figures 6 to 8, both the computation results and measurements with increasing discharge infer a trend that the large vortex enlarged, with its core moving further downstream and its affected area extending in the streamwise direction. The vortex core from numerical results is located

at a higher elevation and more downstream compared to that of measurements. However, the measured flow patterns indicate that the location of the vortex changed slightly with increasing discharge. As the sluice gate intruded deeper (see Figure 8), the horizontal velocity gradient around the gate tip became larger. The measured vortex enlarged in the streamwise and vertical directions and its core slightly moved further downstream. Comparing with numerical results, the same trend over the vortex extension and influenced region was well predicted, but not the precise location of the vortex core. In addition, compared to section Y3 in Figure 8, the numerical results for section Y1 show an obvious deformation of the large vortex due to the suction effect, but the measured patterns do not show this feature. This may be partly due to the insufficient resolution and area limitation of ADV measurement. More specifically, the upper vector field in Figure 8(e) near the pipe does not fully compensate for the shape of the large vortex as presented in the streamline. The full-field measured flow field is more likely to result in the streamline pattern of the numerical result, as shown in Figure 8(b), provided that sufficient spatial resolution is used in the experiment. The streamline pattern may sometimes lead to misunderstanding with limited measured area. Figure 9 compares the simulated flow patterns for case SL and case SE to examine the effect of discharge. The comparison shows that both the strengths and locations of the vortex core are different. In general, the overall pattern gives no further surprise for the Reynolds numbers up to 10^5 .

Figures 10 and 11 show the comparisons of time-averaged and instantaneous flow pattern in section Y3 for case SL and case SLB, respectively. The frame amid the gate and the pipe is shown. The time-elapsd patterns, as shown in Figure 10, show an evolutionary process of transporting and merging between two vortices. Figure 11 gives a more complete process: two separate vortices exist clearly at $0.47T$, and with the small one (left) moving forward toward the large one near the pipe wall with time. At $0.70T$, the two vortices merged into one vortex, and then a new small vortex formed at $0.73T$. The evolutionary process of transporting and merging in all cases is highly unsteady and has no

FIGURE 6: Time-averaged flow patterns on x - z plane for case SL. Numerical results (streamline only) for sections (a) Y1, (b) Y3, and (c) Y5. Measurements (vector and streamline) for sections (d) Y1, (e) Y3, and (f) Y5.

consistent period. The time-elapsed streamline indicates that the flow of the lower region is quite steady and uniform. Of note, the chaotic region extends near the elevation of the bell mouth in case SLB, as shown in Figure 10. This may account for the increasingly undesirable flow with deeper gate submergence, which leads to poor pump efficiency [7, 8, 10, 22].

Figure 12 shows the time-averaged comparisons of numerical and experimental flow patterns in the x - y planes for sections Z2 and Z4, located below and above the suction mouth, respectively. As expected, the overall flow patterns are line symmetric with respect to the $y = 0$ plane. The flow was quite uniform before $x = 800$ mm for both the numerical and experimental results. At $x = 800$ mm, both results show that the flow field begins converging toward the pipe center. Being obstructed by geometry, the measurement of the flow field near the suction pipe or bell mouth is not available. As a result, the numerical model extends the overall

flow pattern. Near the suction pipe, two typical pairs of wall-induced vortices appear in the pattern in section Z4, which agrees with the results reported by Constantinescu and Patel [13]. The calculated streamline corresponding to the suction-pipe center, as shown in Figure 12(a), is slightly asymmetric. This may be induced by nonlinear effects and the disturbance of the sluice gate.

16. Time-Averaged Streamwise and Vertical Velocity Profiles

Figures 13 and 14 show time-averaged streamwise velocity (u) and vertical velocity (w) profiles in section Y3, respectively, for case SL and case SLB. The x -axis and y -axis coordinates are, respectively, normalized by the mean axial velocity of the cylindrical pipe flow (V_p) and the diameter of the cylindrical pipe (D_p). Among sections Y1 to Y5, the largest discrepancy

FIGURE 7: Time-averaged flow patterns on x - z plane for case SH. Numerical results (streamline only) for sections (a) Y1, (b) Y3, and (c) Y5. Measurements (vector and streamline) for sections (d) Y1, (e) Y3, and (f) Y5.

between measurements and numerical results was found for section Y3 for all cases. As a result, profiles in section Y3 are particularly shown and discussed.

As shown in Figure 13, the effect of the intruding gate dominated the upper u profiles. In section X7 (near the front of the bottom step), a subtle detail in vertical velocity profile, around $z/D_p = 0 \sim 0.5$, was well captured and extended by the numerical model. This detail describes the water body being obstructed by the bottom step and being forced to move upward. Comparing the u profiles in sections X7 and X8, there is an overall greater magnitude in section X8 than that in section X7 due to the reduction of flow area. In sections X8 and X10, both the numerical and experimental results reveal that the streamline only slightly converged into the entrance. An examination of sections 14 and 16 clearly shows the strong influences of the cylindrical pipe on both velocity components. Figure 14 shows the velocity profiles for case

SLB. The features of the velocity profiles are almost identical to those of case SL, except for the affected region of the intruding gate.

The dimensionless velocity distributions for the approaching flow Reynolds number, ranging from 4.9×10^4 to 10^5 , have the same patterns and little discrepancy in magnitude. This means that the discharge has little impact on the flow field in the approaching flow. A comparison of velocity profiles for various gate submergence levels is shown in Figure 15. The upper streamwise profiles indicate that flow was obstructed by the sluice gate. Furthermore, the magnitude of the streamwise velocity around the level of the bell mouth became larger with increasing gate submergence. In other words, increasing gate submergence increases the momentum of flow in the pipe. The rise of nonlinearity and turbulence would disturb the impeller operation and further deteriorate the pump efficiency.

FIGURE 8: Time-averaged flow patterns on x - z plane for case SLB. Numerical results (streamline only) for sections (a) Y1, (b) Y3, and (c) Y5. Measurements (vector and streamline) for sections (d) Y1, (e) Y3, and (f) Y5.

17. Swirl Angle

The swirl angle is the most widely used evaluation index of pump efficiency in hydraulic models. A worse approaching flow condition often leads to larger swirl angles. In industrial applications, a commonly acceptable magnitude is 5° [10, 22].

The maximum swirl angle, as shown in Figure 16, grows in proportion to the submergence Froude number and gate submergence. The worst flow condition, with $F_h = 0.56$ and 171 mm gate submergence, yields even three times magnitude than general operation (case SL). However, for the time-averaged swirl angle, shown in Figure 17, the differences caused by the submergence of the sluice gate and the Froude number become much less appreciable in comparison with measurements of the maximum swirl angle. Although a line-symmetric pump sump is considered, the highly unsteady

and nonuniform flow conditions due to the high flow rate and strong turbulence gave rise to undesirable magnitude in swirl angle. Generally, an instantaneous rising swirl angle negatively affects the impeller during long-term operation, even though the probability of occurrence may not be high. Therefore, it is worth noting that the maximum swirl angle should play a confident evaluation index instead of time-averaged measurement.

18. Calculation of Swirl Angle by Numerical Simulation

This study determined the swirl angle from numerical results using a simple mathematical approach. It should be noted that no real swirl meter or moving object was implemented in

FIGURE 9: Comparison of time-averaged flow patterns on x - z plane between case SL and case SE. Note that only numerical results are shown.

the numerical modeling; an imaginary massless swirl meter was used instead. To calculate the swirl angle, the swirl is viewed as a forced vortex in which the flow in the pipe follows one axis of the swirl. A total of 36 cross sections (above the bell-mouth neck) in the pipe were selected from the simulation results. The rotation states of the 36 horizontally sliced discs were calculated. Theoretically, the circulation is defined as the line integral around a closed curve of the fluid velocity and can be related to vorticity by the Stokes theorem, which is written as

$$\Gamma = \oint_c \mathbf{u} \cdot d\mathbf{L} = \iint_s (\nabla \times \mathbf{u}) \cdot d\mathbf{A} \approx \sum_{i=1}^n \omega_{ki} A_i, \quad (19)$$

where Γ is the circulation. ω_{ki} and A_i are, respectively, the vorticity and the area occupied by ω_{ki} . Note that A_i is a circular area with radius $0.75D_p$, which is identical to the radius of the swirl meter (see Figure 3), as shown in Figure 18. Assuming that the pipe flow is a forced vortex or rotational flow, the mean (over circumference) angular velocity (V_θ) of each disc can be evaluated as

$$V_\theta = \frac{\Gamma}{0.75\pi D_p}. \quad (20)$$

Combining (1) and (20) yields

$$\Phi = \tan^{-1} \left(\frac{\Gamma}{0.75\pi D_p V_p} \right). \quad (21)$$

Note that the real pipe flow condition is not uniform in the tangential direction at a radius of $0.75D_p/2$; therefore, (21) is only an approximate expression. This configuration is a compromise; this way the swirl angle can be estimated without the presence of a swirl meter.

The maximum and time-averaged results for cases SL, SLB, and SLA are, respectively, shown in Figures 16 and 17. In Figure 16, the magnitude of the numerical results is quite close to that of the measurements in case SL and case SLA, with errors of about 1.7% and 5.9%, respectively. For case SLB, the error is 22.9%. The magnitudes for case SM and case SH are 3.9° and 4.4° , respectively, corresponding to errors of 15.9% and 21% in comparison with measurements. Figure 17 shows that the numerical results agree well with measurements. The largest error, 30%, was found in case SL. For time-averaged results, both the numerical and experimental results indicate that the time-averaged swirl angle was not sensitive to changes of discharge and the gate submergence

FIGURE 10: Time-averaged and instantaneous flow patterns of streamline in section Y3 for case SL. Note that the blue strip marks the free surface.

level. Although the time-averaged swirl angles varied from 1.0° to 1.8° , the influence of such magnitude on usual pump operation is insignificant. In the experiment, the interaction between the swirl meter and the undesirable flow condition may result in a more complicated flow field that increases the magnitude of the swirl angle. The existence of the swirl meter may thus partly account for the increasing error with increasing gate submergence and discharge. As stated above, the swirl meter was not modeled in the numerical simulation. The deviation between measurements and computed results may arise from neglecting the complex motion of the swirl meter interacting with the fluid flow in the simulation. In

summary, the trend of the computed swirl angles was consistent with that of measurements. The magnitudes for most cases are acceptable. However, for industrial applications, experiments should be used to evaluate pump efficiency via swirl angle.

19. Free Surface

Figure 19 shows the time-averaged and instantaneous patterns of the free surface for case SE. The instantaneous patterns of the free surface within the time-averaged period were examined. The free surface for each case slightly oscillates

FIGURE 11: Time-averaged and instantaneous flow patterns of streamline in section Y3 for case SLB. Note that the blue strip marks the free surface.

with time. The free surface is a flat plane except for the region in the vicinity of the pipe wall. Near the upstream rim of the pipe wall, the fluctuation became more active. The collisions between the flow and the solid pipe wall, the downflow, and the suction force should mainly account for the intensity of the fluctuation. Cases with high gate submergence (cases SLA and SLB) had weaker fluctuation (data not shown here). The inflow velocity dominates the stability of the free surface. A similar phenomenon was described by Chuang and Hsiao [5]. An increase in the Froude number results in stronger fluctuation on the free surface, especially for the water body

near the pipe wall. Although such kind of examination drew no conclusive conclusion to evaluate pump efficiency, this numerical model, TRUCHAS here, did further provide a more realistic simulation and another vision for pump sump problems.

20. Conclusion

Numerical and experimental investigations of pump sump flows with various discharges and gate submergence levels

FIGURE 12: Time-averaged flow patterns on the x - y plane for case SL. Numerical results for sections (a) Z2 and (b) Z4 and measurements for sections (c) Z2 and (d) Z4.

FIGURE 13: Comparison of time-averaged velocity profiles between numerical simulation and measurements in section Y3 for case SL.

FIGURE 14: Comparison of time-averaged velocity profiles between numerical simulation and measurements in section Y3 for case SLB.

FIGURE 15: Comparison of time-averaged velocity profiles in section Y3 among cases SL, SLA, and SLB.

FIGURE 16: Comparison of maximum swirl angle for various submergence levels of sluice gate and various F_h values.

were conducted and the results were compared. The main conclusions of this study are as follows.

- (1) The flow interacting with the sluice gate, bottom step, and intake pipe was described by a streamline pattern and velocity profile. Notably, a large counterclockwise vortex formed in the upper water body between the gate and the suction pipe. The numerical and experimental results indicate that the vortex core moved downstream with increasing discharge. In addition, the vertical range of the vortex extended in the negative z -axis with deeper gate submergence. By examining the evolution of the large vortex through numerical results, it was found that the vortex formation is highly unsteady and has no consistent period for all cases. The vortex deformation caused by the suction effect was well captured by numerical modeling. However, the ADV measurement failed to capture the whole vortex pattern due to its area limitation.
- (2) The velocity profile in the approaching flow was determined using numerical simulation and laboratory measurement. Numerical modeling was quantitatively validated and then applied to examine the flow field where the ADV measurement was not available. The suction effect reflects the velocity profiles close to bell mouth. The x -axis velocity reached $0.4 V_p$ in section X16 around the entrance level. The local Froude number based on the clearance (64 mm) for case SE was calculated as 0.41.

FIGURE 17: Comparison of time-averaged swirl angle for various submergence levels of sluice gate and various F_h values.

- (3) The swirl angle is a straightforward and fairly effective indicator for evaluating pump efficiency. Its magnitude should be as small as possible, at least no larger than 5° . In this study, it was shown that the swirl angle increases with increasing discharge and/or deeper gate submergence. Acceptable value was found using time-averaged swirl angle as an evaluation index. However, the instantaneous data indicated unfavorable magnitudes in most cases. As addressed by Hwang and Yang [10], the sudden rise of the swirl angle with great magnitude may significantly reduce pump efficiency, though this occurrence is not frequent. Numerical results were used to obtain the swirl angle. Instead of modeling a moving object, the vorticity was used to calculate circulation, and then angular velocity was determined by assuming rotational pipe flow. The predicted values agree well with measurements in terms of trend and magnitude. This broadens the application for TRUCHAS for simulating flow with different geometries and layouts.
- (4) By examining the time-averaged and instantaneous free surface, it was observed that fluctuation is obvious only near the pipe wall. The fluctuation intensity increased with increasing discharge but remained invariant with gate submergence.
- (5) Measurements confirmed that TRUCHAS can both qualitatively and quantitatively predict pump sump flows with a complex geometry. However, some phenomena, such as a free surface vortex (air entrainment), cannot be described due to the assumption of

FIGURE 18: Diagram of how to obtain circulation on each selected disc. (a) A horizontally sliced disc (black area) at a given elevation. (b) A disc is divided into several small parts, where each dot indicates vorticity $\omega_{ki} (= \partial u_y / \partial x - \partial u_x / \partial y)$ at its occupied area (A_j).

FIGURE 19: Time-averaged and instantaneous patterns of free surface around pipe for case SE.

a void background. In addition, the addition of the impeller in the simulation should gain more valuable information on cavitation and water-impeller interaction for engineering design. TRUCHAS is able to handle multiphase flow and moving boundaries, but more development and refinement are required since the use of the built-in module is quite limited.

Nomenclature

- A_i : Area of sliced disc
- $\mathbf{A}$: Surface area vector
- C_S : Smagorinsky coefficient
- D_p : Diameter of cylindrical pipe
- F : Cell surface

F_h :	Submergence Froude number
G :	Filter function
$\mathbf{g}$:	Gravitational force vector
g_i :	i th component of gravitational field
H :	Mean water depth
h_s :	Intake submergence
ℓ_S :	Smagorinsky length scale
N :	Revolutions per second
n :	n th time step
$\mathbf{n}$:	Interfacial normal vector
$\bar{P}$:	Filtered pressure
Q :	Discharge
R_a :	Approaching flow Reynolds number
S :	Source term
$\bar{S}_{ij}$:	Filtered rate of strain
t :	Time
T :	Time interval for time-averaged data
$\mathbf{u}$:	Velocity vector
$\bar{u}_i$:	i th component of filtered velocity field
u_x :	Velocity in x -axis
u_τ :	Shear velocity
u_y :	Velocity in y -axis
V_a :	Mean streamwise velocity of approaching inlet flow
V_b :	Mean axial velocities at the bell mouth
V_P :	Mean axial velocity
V_Φ :	Tangential velocity
$\bar{V}_\theta$:	Angular velocity of a sliced closed circle
W :	Width of approaching channel
y_w :	Distance from wall to center of first cell adjacent to wall
Γ :	Circulation
Δt :	Time step increment
Δ :	Filter width
δ_{ij} :	Kronecker delta operator
ς :	Measure of cell size
κ :	von Karman constant
ν :	Kinematic viscosity
ν_t :	Sub-grid-scale eddy viscosity
ν_w :	Near-wall eddy viscosity
ρ_m :	Density of m th fluid
$(\bar{\tau}_{ij})_l$:	Filtered molecular viscous stress tensor
$(\bar{\tau}_{ij})_t$:	Subgrid scale Reynolds stress tensor
Φ :	Swirl angle
ϕ :	Arbitrary quantity
ω_k :	k th component of vorticity
V :	Cell volume
Superscript $[*]$:	Intermediate level
Subscript $[f]$:	Cell surface
Subscript $[F]$:	Solenoidal face.

Conflict of Interests

The authors declare that there is no conflict of interests regarding the publication of this paper.

Acknowledgments

The authors wish to thank the National Science Council of Taiwan for financially supporting this research under Grants NSC 101-2628-E-006-011 and 101-2628-E-006-015-MY3. The authors would also like to thank Mr. Chun-Po Lin and Professor Tso-Ren Wu for their help.

References

- [1] M. Ansar, T. Nakato, and G. Constantinescu, "Numerical simulations of inviscid three-dimensional flows at single- and dual-pump intakes," *Journal of Hydraulic Research*, vol. 40, no. 4, pp. 461–470, 2002.
- [2] H.-X. Chen and J.-H. Guo, "Numerical simulation of 3-D turbulent flow in the multi-intakes sump of the pump station 1 Project supported by the Shanghai Leading Academic Discipline Project (Grant No: Y0103)," *Journal of Hydrodynamics*, vol. 19, no. 1, pp. 42–47, 2007.
- [3] A. Skerlavaj, F. Vehar, R. Pavlin, and A. Lipej, "A hydraulic study of cooling water intake structure," in *Proceedings of the 3rd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems*, Brno, Czech Republic, 2009.
- [4] M. Ansar and T. Nakato, "Experimental study of 3D pump-intake flows with and without cross flow," *Journal of Hydraulic Engineering*, vol. 127, no. 10, pp. 825–834, 2001.
- [5] W.-L. Chuang and S.-C. Hsiao, "Three-dimensional numerical simulation of intake model with cross flow," *Journal of Hydrodynamics*, vol. 23, no. 3, pp. 314–324, 2011.
- [6] G. Arboleda and M. El-Fadel, "Effects of approach flow conditions on pump sump design," *Journal of Hydraulic Engineering*, vol. 122, no. 9, pp. 489–494, 1996.
- [7] M. J. Prosser, "The hydraulic design of pump sumps and intakes," 1977.
- [8] Hydraulics Institute Standards, *Centrifugal, Rotary and Reciprocating Pumps*, HIS, Cleveland, Ohio, USA, 14th edition, 1998.
- [9] M. C. Quick, "Efficiency of air-entraining vortex formulation," *Journal of the Hydraulics Division ASCE*, vol. 96, no. 7, pp. 1403–1416, 1970.
- [10] K. S. Hwang and C. H. Yang, "Hydraulics model testing of circulating-water pump sump in Shen-Ho power plant," Research Report 293, Taiwan Hydraulics Laboratory, Tainan, Taiwan, 2003.
- [11] J. P. Tullis, "Modeling in design of pumping pits," *Journal of Hydraulic Engineering: (ASCE)*, vol. 105, no. 9, pp. 1053–1063, 1979.
- [12] C. E. Sweeney, R. A. Elder, and D. Hay, "Pump sump design experience: summary," *Journal of Hydraulic Engineering, ASCE*, vol. 108, no. 3, pp. 361–377, 1982.
- [13] G. S. Constantinescu and V. C. Patel, "Numerical model for simulation of pump-intake flow and vortices," *Journal of Hydraulic Engineering*, vol. 124, no. 2, pp. 123–134, 1998.
- [14] S. Li, Y. Lai, L. Weber, J. M. Silva, and V. C. Patel, "Validation of a three-dimensional numerical model for water-pump intakes," *Journal of Hydraulic Research*, vol. 42, no. 3, pp. 282–292, 2004.
- [15] T. E. Tokyay and S. G. Constantinescu, "Validation of a large-eddy simulation model to simulate flow in pump intakes of realistic geometry," *Journal of Hydraulic Engineering*, vol. 132, no. 12, pp. 1303–1315, 2006.

- [16] W. Yulin, L. Yong, and L. Xiaoming, "PIV Experiments on Flow in a Model Pump Suction Sump," Research Rep, Thermal Engineering Department, Tsinghua University, Beijing, China, 2000.
- [17] M. Padmanabhan and G. E. Hecker, "Scale effects in pump sump models," *Journal of Hydraulic Engineering*, vol. 110, no. 11, pp. 1540–1556, 1984.
- [18] A. K. Jain, K. G. Ranga Raju, and R. J. Garde, "Vortex formation at vertical pipe intakes," *Journal of Hydraulic Engineering*: (ASCE), vol. 104, no. 10, pp. 1429–1445, 1978.
- [19] P. J. Rusello, A. Lohrmann, E. Siegel, and T. Maddux, "Improvements in acoustic doppler velocimetry," in *Proceedings of the 7th International Conference on Hydropscience and Engineering (ICHE '06)*, September 2006.
- [20] G. Voulgaris and J. H. Trowbridge, "Evaluation of the acoustic Doppler velocimeter (ADV) for turbulence measurements," *Journal of Atmospheric and Oceanic Technology*, vol. 15, pp. 272–289, 1998.
- [21] D. G. Goring and V. I. Nikora, "Despiking acoustic doppler velocimeter data," *Journal of Hydraulic Engineering*, vol. 128, no. 1, pp. 117–126, 2002.
- [22] M. Ansar, *Experimental and theoretical studies of pump-approach flow distributions at water intakes [Ph.D. thesis]*, University of Iowa, Iowa City, Iowa, USA, 1997.
- [23] V. P. Rajendran, S. G. Constantinescu, and V. C. Patel, "Experimental validation of numerical model of flow in pump-intake bays," *Journal of Hydraulic Engineering*, vol. 125, no. 11, pp. 1119–1125, 1999.
- [24] Y. B. Li, *Effect of non-uniform clearness of water screens on pump intake flow [M.S. thesis]*, National Cheng Kung University, Tainan, Taiwan, 2006 (Chinese).
- [25] T. R. Wu, *A numerical study of three-dimensional breaking waves and turbulence effects [Ph.D. thesis]*, Cornell University, Ithaca, NY, USA, 2004.
- [26] S. C. Yim, D. T. Cox, and M. M. Park, "Experimental and computational activities at the oregon state university NEES Tsunami Research Facility," *Science of Tsunami Hazards*, vol. 28, no. 1, pp. 1–14, 2009.
- [27] K. C. Hu, S. C. Hsiao, H. H. Hwung, and T. R. Wu, "Three-dimensional numerical modeling of the interaction of dam-break waves and porous media," *Advances in Water Resources*, vol. 47, pp. 14–30, 2012.
- [28] Y. T. Wu, C. L. Yeh, and S. C. Hsiao, "Three-dimensional numerical simulation on the interaction of solitary waves and porous breakwaters," *Coastal Engineering*, vol. 85, pp. 12–29, 2014.
- [29] G. Ma, M. Shi, and J. Kirby, "A polydisperse two-fluid model for surf zone bubble simulation," *Journal of Geophysical Research*, vol. 116, Article ID C05010, 2011.
- [30] T.-R. Wu, C.-J. Huang, M.-H. Chuang, C.-Y. Wang, and C.-R. Chu, "Dynamic coupling of multi-phase fluids with a moving obstacle," *Journal of Marine Science and Technology*, vol. 19, no. 6, pp. 643–650, 2011.
- [31] P. L.-F. Liu, T.-R. Wu, F. Raichlen, C. E. Synolakis, and J. C. Borrero, "Runup and rundown generated by three-dimensional sliding masses," *Journal of Fluid Mechanics*, vol. 536, pp. 107–144, 2005.
- [32] J. W. Deardorff, "Numerical study of three-dimensional turbulent channel flow at large reynolds numbers," *Journal of Fluid Mechanics*, vol. 41, no. 2, pp. 453–480, 1970.
- [33] J. Smagorinsky, "General circulation experiments with the primitive equations: I the basic equations," *Monthly Weather Review*, vol. 91, pp. 99–164, 1963.
- [34] W. Cabot and P. Moin, "Approximate wall boundary conditions in the large-eddy simulation of high Reynolds number flow," *Flow, Turbulence and Combustion*, vol. 63, no. 1, pp. 269–291, 2000.
- [35] M. Vinokur, "An analysis of finite-difference and finite-volume formulations of conservation laws," *Journal of Computational Physics*, vol. 81, no. 1, pp. 1–52, 1989.
- [36] J. H. Ferziger and M. Peric, *Computational Methods for Fluid Dynamics*, Springer, 1996.
- [37] A. J. Chorin, "Numerical Solution of the Navier-Stokes equations," *Mathematics of Computation*, vol. 22, pp. 745–762, 1968.
- [38] A. J. Chorin, "On the convergence of discrete approximations of the Navier-Stokes equations," *Mathematics of Computation*, vol. 23, pp. 341–353, 1969.
- [39] C. W. Hirt and B. D. Nichols, "Volume of fluid (VOF) method for the dynamics of free boundaries," *Journal of Computational Physics*, vol. 39, no. 1, pp. 201–225, 1981.
- [40] J. Gullbrand and F. K. Chow, "The effect of numerical errors and turbulence models in large-eddy simulations of channel flow, with and without explicit filtering," *Journal of Fluid Mechanics*, vol. 495, pp. 323–341, 2003.