Extent, capacity and possibilities of computational fluid dynamics as a design tool for pump intakes: a review

Jie Zhang and Tien Yee

ABSTRACT

Flow near pump intakes is three-dimensional in nature, and is affected by many factors such as the geometry of the intake bay, uniformity of approach flow, critical submergence, placements and operation combinations of pumps and so on. In the last three decades, advancement of numerical techniques coupled with the increase in computational resources made it possible to conduct computational fluid dynamics (CFD) simulations on pump intakes. This article reviews different aspects involved in CFD modeling of pump station intakes, outlines the challenges faced by current CFD modelers, and provides an attempt to forecast future direction of CFD modeling of pump intakes.

Key words | computational fluid dynamics, design, pump intake, turbulent model, vortex

Jie Zhang (corresponding author)
Carollo Engineers, Inc.,
1218 Third Ave, Suite 1600,
Seattle, WA 98101,
USA
E-mail: *jiezhang.cfd@gmail.com*

Tien Yee
Civil and Construction Engineering, Kennesaw State University,
1100 South Marietta Parkway,
Marietta, GA 30060,
USA

INTRODUCTION

Pump intake is a system of withdrawing water from a river or a reservoir, commonly used to supply water for water treatment plants, irrigation systems, or electric-power generating plants. A pump intake system usually includes a channel which connects water source (i.e. river or reservoir) and pump and flow training devices such as floor and sidewall splitters, flow dividers and turning vanes directing the flow (Tokyay & Constantinescu 2005a). Sometimes a pump intake system also includes a concrete-constructed pump bay (i.e. sump) which is often used as an intermediate buffer to control turbulence intensity and uniformity of the inflow (ANSI/HI 9.8 1998). Two fundamental types of flow problems that significantly affect the efficiency of pump intake systems are air entrainment problems in the pump sump and sediment problems at or near the intakes (Issa *et al.* 2008). Both problems are induced by vortices in the system, making vortices important, but an undesirable characteristic of a pump intake system. Vortices in a pump intake system are usually classified as free surface or subsurface vortices based on the locations where they originate, either on the water surface or subsurface. Vortices can be further classified according to the strength of the swirl or vorticity (visual observation and dye tracer test) and observable development of air core, please refer to the classifications in Figure 1.

Poor geometric configuration design is the primary reason for the emergence of various vortices. These vortices may cause the loss of hydraulic head, decrease in flow and efficiency and, when excessive, may cause cavitation problems. All these consequences will have a negative impact on the longevity of pumps.

To mitigate these problems, it is beneficial for engineers, researchers and pump manufacturers to study the flow near the pump intakes to identify the environment that induces vortices. Traditionally, pump intakes were designed based on the recommended design criteria from the Hydraulic Institute and then tested using scaled models to ensure a problem free environment within the intake system, especially near pump intakes. However, scaled models are expensive to construct and is site specific. Further, testing

doi: 10.2166/ws.2018.004

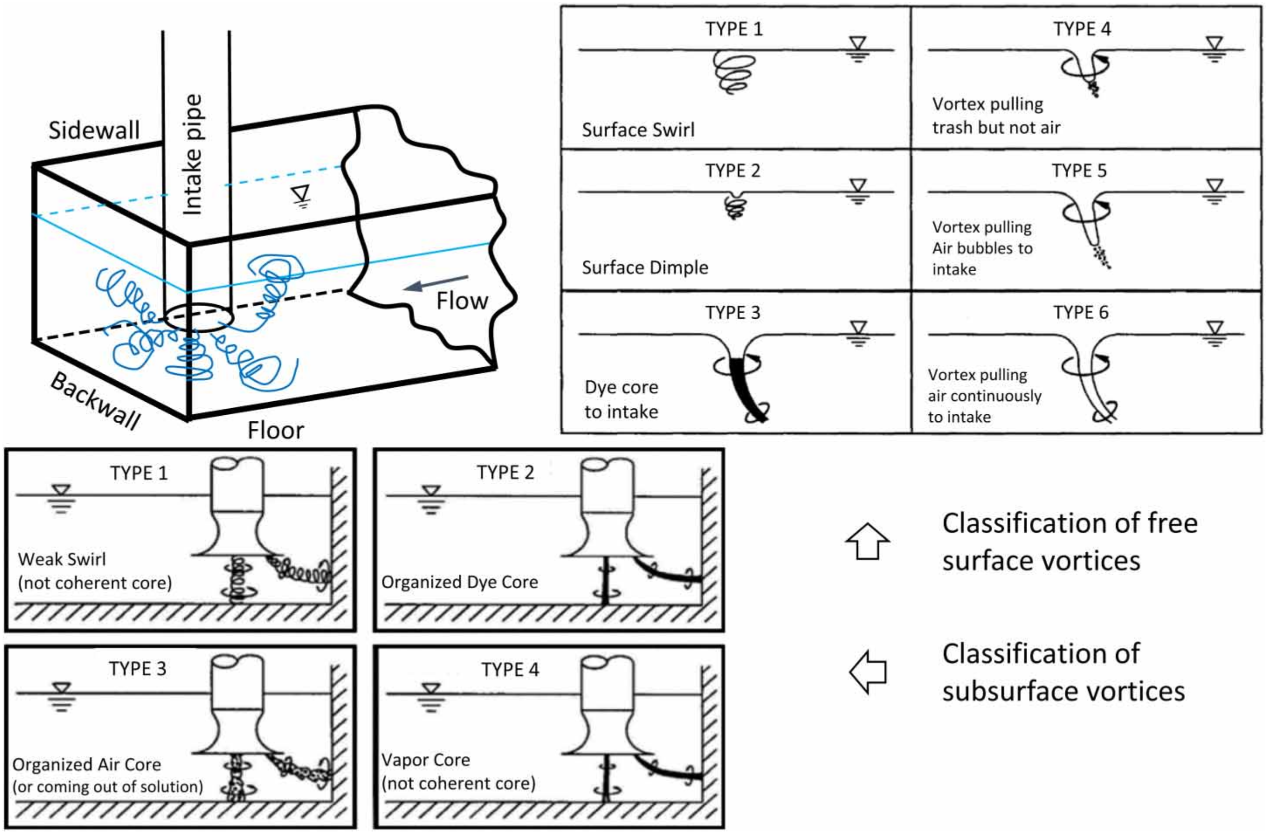

Figure 1 | Schematics of a typical pump intake system and classification of vortices (the difference between free surface and subsurface vortices is the location where they originate) (adapted from ANSI/HI 9.8 2012).

for all combinations of operating cases is labor intensive and time consuming. The other alternative for studying fluid flow near pump intakes is through the use of an advanced numerical technique also known as the computational fluid dynamics (CFD), which had been widely applied in other disciplines and areas such as aerospace and aeronautics (Fujii 2005; Gao *et al.* 2013), environmental hydraulics (Bates *et al.* 2005), oceanography (Tejada-Martinez & Grosch 2007) and other physical sciences (Blazek 2015). CFD studies of pump intake systems have been uncommon until the past two decades (Tokyay & Constantinescu 2005a). The phenomenon is similar in other areas of environmental engineering, such as water disinfection (Zhang *et al.* 2014). Since CFD can offer pump station designers the ability to study flow near pump intakes of varying geometries without the high cost of scaled models, it would be interesting to present researchers and engineers a fair appraisal of CFD performance in modeling flow in pump intakes.

There are two primary goals that this article will attempt to address. These goals are:

(1) to reveal the applications of CFD in understanding the flow structures in pump intake systems;
(2) to evaluate the capacity of CFD in troubleshooting and design of pump intake systems

With these goals in mind, the following sections will review previous work done on developing and improving CFD models for flows in pump intakes; and then review and summarize the findings associated with pump intake designs using CFD.

DEVELOP AND REFINE TOOLS

As pointed out by Rajendran *et al.* (1999), there are two issues that need to be addressed before CFD can be effectively applied to practical pump intake systems. Firstly, CFD models have to be more robust to handle the complex geometry of pump intakes, such as multiple pumps, partition walls, corner fillets, screens, and other flow-training devices. Secondly, CFD models have to be improved to more accurately capture complex hydrodynamic phenomena such as turbulence, unsteadiness of flows, and free surface effects. Inability to resolve either or both of the mentioned issues could result in a significant discrepancy between CFD predictions and physical experimental results. The advancement of meshing tools has reduced the complication faced in modeling of complex geometries. Even though the meshing process can sometimes be time-consuming, advancement in meshing tools enable CFD to be used for more complex geometries. The remaining primary sources of the discrepancies stem from assumptions made by CFD modelers such as the steady flow assumption, eddy-viscosity assumption, and single-phase flow assumption. Since the first couple of assumptions are dependent upon the selection of a turbulence model, it will be first discussed in the following sections.

Turbulence models

There are three primary and well-known turbulent flow modeling strategies (Pope 2000), namely the Reynolds-averaged Navier–Stokes simulation (RANS), large eddy simulation (LES), and direct numerical simulation (DNS). DNS resolves the governing Navier–Stokes equations numerically over the entire range of turbulent scales. Because of this reason, DNS requires very fine spatial meshes and temporal stepping. The requirements on temporal and spatial mesh resolution demands prohibitively high computational resources, rendering it unsuitable for most practical engineering applications.

RANS is a statistical approach for the simulation of turbulent flow. RANS involves the application of Reynolds averaging to decompose Navier–Stokes equation solution variables into their means and the turbulent fluctuations about these means. The Reynolds-averaged equations are then solved for the mean component of the flow without explicit computation of the turbulent scales. Instead, the Reynolds-averaged equations contain Reynolds stress terms accounting for the effect of the unresolved turbulent scales on the explicitly computed mean component of the flow. Often, the Reynolds stresses are modeled following the eddy viscosity hypothesis (Pope 2000). The primary advantage of a RANS type model is its relatively low requirement on computing resources given that it only resolves the mean flow. The RANS model is, therefore, a good candidate for applications in pump intake studies, particularly when simulating cases with steady flow.

LES aims to reduce the requirements on mesh resolution imposed by DNS. The idea of LES is to use a spatial filter to separate the turbulent flow field into two components. The larger scale, more energetic structures that can be resolved by the numerical method on a given mesh are referred to as the resolved scales. The smaller structures that cannot be captured by the mesh are called sub-grid scales. The influence of sub-grid scales on resolved scales must be modeled. Often, the sub-grid-scale (SGS) model is also based on an eddy viscosity hypothesis. The principle of LES is based on the fact that the small (unresolved) scales of turbulence are homogeneous and isotropic and are therefore easier to model compared to the larger scales. Furthermore, these small (unresolved) scales are universal and thus allow different SGS models to be used for different flow problems. Results of LES would therefore be closer to that of the DNS under fine mesh as the size of scales that require modeling become smaller and less energetic. LES lies between DNS and RANS in terms of accuracy and computational cost. LES is more affordable than the DNS in terms of computational requirements, but is still computational demanding compared to RANS. Nonetheless, LES is still a candidate for modeling of pump intakes.

Among the three strategies discussed above, RANS was the most commonly used approach for modeling flow in pump intakes, mainly due to its low requirements on computing resources. LES only became more affordable due to the rapid advancement of computing technology in recent years.

One of the earliest works on RANS simulation of pump intake referenced the work done by Tagomori & Gotoh (1989). They employed the Finite Volume approach to solve the RANS equations and the k-ε turbulent closure model. Their work had been cited and mentioned by a couple of different researchers (Constantinescu & Patel 1998; Issa *et al.* 2008, 2009). Tagomori & Gotoh (1989) studied the effects of velocity distribution in approach channel on the formation of vortices. Their work primarily focused on the effect of non-uniform inlet flow on vortex formation and the effects of additional devices to prevent vortex formation (Tagomori & Gotoch 1989). They found that unlike the vertical plane velocity, non-uniformity in the horizontal plane velocity has significant effects on the pump performance. The emphasis of this paper was geared towards the generation mechanism of surface and subsurface vortices and several preventative mechanisms. The article concluded that mitigation strategies for vortices should be based on the flow pattern and the vortex generation mechanism. The article also further pointed out that the flow pattern in an approach channel can be studied using CFD analysis. While not many details were available about their numerical model, nonetheless, the authors of this article acknowledge the importance of Tagomori and Gotoh's work, as it is among the earliest dated publication on CFD of pump intakes simulation employing RANS and k-ε turbulent closure model and one of the pioneering CFD simulations of pump intakes.

Takata *et al.* (1992) studied submerged vortices near pump intakes using LES. Their work studied intake flows at Reynolds number from $1.5–5.5 \times 10^4$. The primary focus of their work was to study the flow field around pump intake, particularly on the formation of submerged vortices. Unfortunately, the grid spacing employed was too coarse for an LES simulation of such complex flow (Constantinescu & Patel 1998). The course grid spacing that was used may have been due to the limitation of computing power at the time. As accessibility, affordability and computing power increases, more CFD research and work were made possible. During the late 1990s pump intake simulations began to garner more attention. Several other research groups attempted to numerically simulate pump intakes using RANS and LES technique with different turbulent and wall models.

Constantinescu & Patel (1998) developed a CFD model to solve the RANS equations with a two-layer k-ε turbulent model. In their work, they chose a simple rectangular sump geometry and a vertical upward pump intake in demonstrating the use of his numerical model to predict the strength and location of free surface and wall-attached vortex. They employed a grid consisting of 550,000 points for the simulation and calculations were performed for a Reynolds number of up to 60,000. Their CFD model was able to predict the location, size and strength of vortices with sufficient details. Constantinescu and Patel's work is significant because ample information was provided such that the geometry of the pump intake could be reproduced if desired. This then allowed other researchers to follow suit in CFD pump intake simulations.

In 1999, another application of CFD for simulation of pump intakes was observed (Roberge 1999) for sump and intake geometry similar to Constantinescu & Patel (1998). Roberge (1999) used a commercial software, FIDAP, to perform all simulations. FIDAP was developed based on the RANS turbulence strategy with various closure models available. Similar to Constantinescu and Patel's work, Roberge (1999) chose the standard k-ε closure model. The findings from this work were complementary to that of Constantinescu & Patel (1998). This work demonstrated that CFD simulation for intake design can be done at a basic level using commercially available software and that CFD applications are not solely performed in academia and research.

Constantinescu & Patel (2000) continued with their prior work by performing another study on pump intake modeling. This time, the primary goal was to observe the sensitivity of outcomes of pump intake simulations for different turbulent closure models. The two turbulent models chosen in the study were the k-ε closure model with wall functions and the k-ω closure models. Three variants of the k-ω models were used, and they are the low-Reynolds number form, the high-Reynolds number form with smooth wall and the high-Reynolds number form with rough wall. Constantinescu & Patel (2000) believed that the two-equation models chosen were economical and if necessary can be improved depending upon a non-linear wall model that they were developing at that time. Approximately 550,000-point grids were used in their simulations and the Reynolds number for the simulation was 120,000 calculated based on the diameter of the pump intake and the pump column velocity. In general, the authors found that the k-ε model was unable to predict the smaller free surface vortices near the pump intake walls, however, all forms of studied closure models were able to predict the main vortices. All forms of turbulent closure models were able to capture the subsurface vortex with slight variation in vortex positions and strengths. It is also interesting to mention that while performing the study, the authors found that weaker vortices were produced when wall roughness is considered in their simulations, suggesting the possibility of using wall roughness as vortex suppression or damper.

Okamura & Kamemoto (2005) assembled a committee consisting of 10 committee members from university faculty members and engineers in Japan to compare the results of pump intake simulations using different commercial CFD software such as ANSYS CFX, STAR-CD, Scryu/Tetra, and ANSYS FLUENT, as well as several in-house codes. A total of 10 cases of different software, numerical techniques, models and grids were simulated. The numerical methods used were the Vortex method, Finite Element Method, Finite Volume Method and Finite Difference Method. The different turbulent closure models used includes the k-ε model, the re-normalization group (RNG) k-ε model and the shear stress transport (SST) model. Table 1 details the different work done with the various numerical methods, difference schemes, turbulent models, number of grids and methods used for vortex identifications.

It is worthwhile to mention that the majority of the contributors in this work (Okamura & Kamemoto 2005) that used a closure model have chosen the k-ε turbulent model or the RNG k-ε model. The reason for consensus in the choice of turbulent closure model was not disclosed in the article. It may be that the k-ε type turbulent model had previously shown to have some success in pump intake simulations such as shown that mentioned earlier (Tagomori & Gotoch 1989; Constantinescu & Patel 1998). While Okamura & Kamemoto (2005) provided insights about the capabilities of the different methods and commercial software packages. These authors also cautioned that the simulation and reported results were based primarily on the benchmark test case which has not been confirmed experimentally. The study supported the fact that the flow characteristics and pattern near the pump bell entrance were not remarkably different between all test cases, however the predicted vortex strength varies between the different methods used to study them.

Tokyay & Constantinescu (2006) simulated a pressurized pump intake using more realistic intake geometry. The pump intake geometry consists of a simple vertical

Table 1 | Characteristics of CFD codes from Okamura & Kamemoto (2005)

Code symbol	A	B	C	D	E	F	G	H	I	J
Code name	Virtual Fluid System 3-D	In-house code	CFX 5.6	CFX 5.6	STAR-CD	STAR-CD	Scryu/Tetra	Scryu/ Tetra 5.0	FLUENT	In-house code
Numerical method	Vortex method	FEM	FVM	FVM	FVM	FVM	FVM	FVM	FVM	FDM
Difference scheme	None	Third order TVD-MUSCL	Second order upwind	Second order upwind	MARS	MARS	MUSCL method	MUSCL method	Second order upwind	CIP
Turbulence model	None	None	k-ε	k-ε	RNG k-ε	RNG k-ε	k-ε	SST	k-ε	None
Number of grids	13,000	580,000	600,000	1,020,000	180,000	1,920,000	330,000	990,000	980,000	330,000
Evaluation method	streamline, vorticity, pressure at vortex core	streamline, vortex centerline	streamline, vorticity, pressure at vortex core	velocity vector, pressure, vorticity	Flow and vortex core line	Pressure at vortex core	Flow pattern and vortex core	vortex core line	velocity vector, vorticity, vortex centerline	Velocity distribution, vorticity

FEM: finite element method; FVM: finite volume method, FDM: finite difference method, SST: shear stress transport, TVD-MUSCL: total variation diminishing monotonic upwind scheme for conservation laws.

pump intake and two identical approaching channels leading to the intake section. In the paper, the LES solver was used along with the Dynamic Smagorinsky turbulent closure model. A mesh of nearly 5 million cells were used in the LES simulation. For the sake of comparison, the authors also simulated the same pump intake using the RANS solver in FLUENT with approximately 1.5 million cells using the SST turbulent model. Tokyay & Constantinescu (2006) then qualitatively and quantitatively compared the two models in predicting the submerged vortex location and properties. In the paper, the authors observed that, while the RANS model was able to produce satisfactory streamlines and indicated the general vortex location, the LES model is able to capture more accurately the position and structure of main sidewall attached vortices, as well as secondary eddies shed from the main one. Further, the authors also found that for their case study, LES was able to predict the turbulent kinetic energy and velocity magnitude distribution better than RANS. The paper also provided subtle hints about: (1) the important role that was played by the numerical technique and turbulent closure model in the modeling of pump intakes and (2) the foreseeable challenges that CFD modelers have to overcome to gain recognition as a reliable tool for industry application.

At about the same time, Li *et al.* (2006) simulated the vertical pump intake of Ansar (1997) using a RANS solver and the standard k-ε turbulent model with wall function. Approximately 1.45 million cells were used to construct the computational domain. The study was done to observe the main flow features in the flow domain and to compare the velocity distribution in the approaching channel and inside the pump column. Experimental and numerical tests were carried out to compare results of two incoming flow conditions, namely inflow condition with 'no-cross flow' and with 'cross flow'. Their CFD simulations was able to reproduce the general flow patterns as well as weak and strong swirls around the suction bell for the 'no cross flow' case scenario. It was also shown in their numerical model that flow at the pump throat agrees reasonably well with experiment, but at the region closest to the back wall, the velocity distribution at the pipe centerline was overestimated and the velocity nearer to the intake pipe wall was underestimated. For the 'cross flow' case scenario, the flow patterns were significantly different and the numerical model was able to predict the general swirls and flow patterns in the flow domain. Within the intake pipe, the numerical model predicted velocity and swirl that did not match the experiment data. The discrepancy between the experimental data and the numerical simulation may be due to the turbulent model used or that the numerical scheme attempted to seek a steady solution whereas the 'cross flow' case (Li *et al.* 2006) contains features that are vastly unsteady. This study shows that CFD was able to capture the important flow features and compared fairly well with experimental data for both test cases, but near the pump throat further investigation into the different options for flow solvers and turbulent models may be necessary.

The issue with the k-ε turbulent model in predicting non-symmetrical and unsteady flow was also pointed out in another paper (Issa *et al.* 2008). Issa *et al.* (2008) conducted numerical tests of flow near pump intake, similar to that in Constantinescu & Patel (1998). Issa *et al.* (2008) used RANS with two different turbulent closure models, the k-ε turbulent model and the k-ω turbulent model, with different submergence depths and inlet boundary layer conditions. The paper reported that the results obtained using k-ε turbulent closure model for the case of low submergence and low water level in the sump, may be erroneous and is supported by the fact that the convergence history was observed to be oscillatory and unstable. The authors also mentioned that for cases with strong vortices, both due to low submergence and thick inlet boundary layer, the k-ω turbulent model would perform better with a stable convergence history. The literature concluded that, for the simulated case study, the k-ω model seems to be the most appropriate turbulent closure model which produces results that better describe a symmetrical flow field. The authors also suggested that more numerical tests are needed as non-symmetrical results for a symmetrical geometry could also have been attributed to the unstructured grids distribution being non-symmetrical in the horizontal plane.

Issa *et al.* (2009) continued their study on pump intake simulation by using the k-ω turbulent model. The authors found that the incoming flow pattern near an intake is sensitive to not only the mesh size and turbulent closures but also the slight change of the inlet flow conditions, which was earlier pointed out in Tagomori & Gotoch (1989). The work amplified the fact that more studies are necessary to clarify the fitting technique and turbulent closure, grid spacing and inlet flow conditions that are suitable for pump intake simulations, especially when the goal is to replicate experimental observations.

Škerlavaj *et al.* (2011) attempted to address the turbulent closure model issues mentioned above and repeated the simulation of the vertical pump intake. The geometry similar to (Wu *et al.* 2000) and (Tokyay & Constantinescu 2006) was used. ANSYS CFX was chosen to be the software used for all simulations performed in this study. Twelve different simulations were performed with different turbulent models and closure models such as the SST, scale-adaptive simulation (SAS) model, Reynolds stress model (RSM), explicit algebraic Reynolds stress model (EARSM), detached eddy simulation (DES) and LES. The curvature correction (CC) option within ANSYS CFX was also assessed for the SAS and SST type closure model. The primary interest of the work was to find a turbulent model that best capture the essences of transient flow and at the same time being computationally affordable for industrial application. The findings from their study was that all the different turbulent models successfully predicted the formation of vortex, however, all the tested steady state models under-predicted the turbulent kinetic energy for transient flows. The baseline explicit algebraic Reynolds stress model (BSL EARSM) and Speziale–Sarkar–Gatski (SSG) RSM models were slow to converge and only recommended for coarse mesh setup. The DES was found to give misleading results for large time steps and to have the similar computation requirement compared to the SAS-CC model. With further tests and comparison to experimental data, the authors recommended that the SAS-CC model is suitable for industrial pump intake simulation even without a supercomputer.

While many CFD simulations of vertical pump intake have been mentioned, CFD have also been applied to horizontal pump intake systems (Pradeep *et al.* 2012). Pradeep *et al.* (2012) presented a case study of CFD application to horizontal pump intake using FLUENT. The work was done to evaluate the capability of CFD to analyze flow field for horizontal pump intakes. The geometry of the intake consists of an approach channel, a transitional slope, a forebay and six horizontal intake pumps. The pump intake was analyzed for four different cases defined by the three operating flow rates and several combinations of running pumps. The RANS solver with the standard k-ε closure model was employed in their simulation. Experiments corresponding to the test cases were also performed to validate their CFD results. The work focuses on the prediction of swirl angles and velocity distributions within the approach channel, as well as near the entry to the draft tube of the intakes. Their finding was positive and encouraging, in that the CFD predictions match reasonably with that of the experiments in terms of velocity distribution at the draft tube entrance, swirl data and circulation within the forebay. The work showed that the CFD model used in their study was able to predict large scale flow circulations, however, may not be able to predict details such as the strength, size and air entraining vortices. In their concluding remarks, the authors acknowledged that the work can be extended to using different numerical models, grid spacing and turbulent models for better prediction of vortices in the sump chambers.

By plotting the Reynolds numbers of the flows studied in the literature above over time, see Figure 2, interesting phenomena can be observed: (1) the overall trend of flow Reynolds number studied using RANS is increasing as expected since the affordable power of computing is increasing; (2) the use of LES is still rare due to its relatively high computational cost. Note that the computational cost of LES is proportional to the Reynolds number. For example, Tokyay & Constantinescu (2005a, 2005b) simulated the

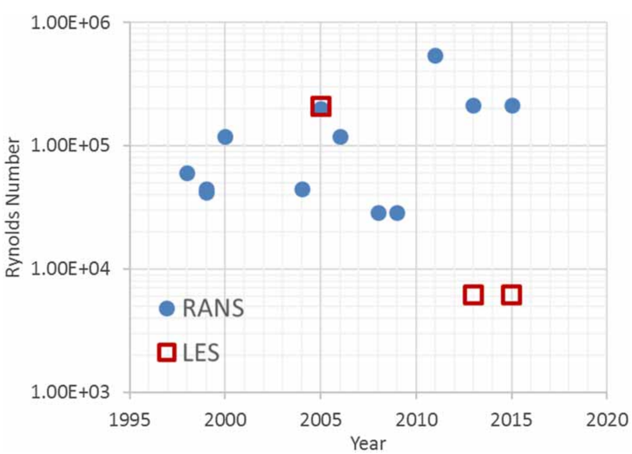

Figure 2 | The Reynolds number of flow in pump intakes studied by CFD versus year.

flow in pump intake system (the Reynolds number is around 2.1×10^5) using LES but at an extremely high cost (i.e. the mesh consist of 5 million cells in total; one LES run consumed 30 days of computing time on 24 processors).

Besides LES and RANS, a relatively new approach, Lattice Boltzmann Method (LBM), was applied in simulating flow in pump intakes in recent years (Schneider *et al.* 2013, 2015). LBM has the advantages of handling complex geometries without manual meshing and a high efficiency in calculating transient flows. However, since only a few commercial softwares were developed, this method was not used as often as RANS and LES.

Free surface effects

Thus far, most of the mentioned literatures have not dealt with the effects of free surface on the formation of surface vortices. In order to incorporate the effects of free surface, a two-phase flow model must be employed. Since a two-phase flow model is more complicated and computational-intensive than a single-phase flow model, attempts of multi-phase flow simulation did not appear until the recent decade. The volume of fluid (VOF) method is the most commonly used two-phase model for pump intakes. The VOF models in commercial software, such as ANSYS CFX (Shukla & Kshirsagar 2008; Matsui *et al.* 2014), ANSYS FLUENT (Bayeul-Lainé *et al.* 2010), STAR CCM + (Bayeul-Lainé *et al.* 2010, 2012), and in-house codes, such as Truchas developed by the Los Alamos National Lab (LANL) (Chuang & Hsiao 2011), have been used to capture the effects of free surface. The results from these studies demonstrated that VOF is able to capture the surface flow in sufficient details to identify the location, size, and strength of the free surface vortices, recall the concept of free surface vortices in Figure 1. Although prediction accuracy can be improved, as mentioned above, employing a two-phase flow model may increase computational burden as well. Samsudin *et al.* (2015) pointed out that balance between computational time and resources versus the degree of desired accuracy have to be achieved for effective application of CFD for pump intakes. In some circumstances, a single-phase flow model may be inadequate to model surface vortices and may lead to erroneous results. An example of such scenario was described in Kim *et al.* (2015). Kim *et al.* (2015) found that single-phase simulations could lead to incorrect predictions near the bell mouth in the cases with an anti-vortex device installed. Such cases may warrant the use of a two-phase flow model such as the VOF. The authors of this article believe that a two-phase flow model such as the VOF may be necessary when signs of surface vortex are showing near the pump intake section or when simulating cases with higher probability of surface vortex formation (i.e. simulation of low submergence cases, expected strong near surface velocity circulation pattern, etc.).

To further increase the prediction accuracy, VOF method was combined with LES strategy (instead of RANS) to capture the transient behaviors of free surface (Zhao & Nohmi 2012). Zhao & Nohmi (2012) noted that while some vortices were successfully predicted, the CFD model have also predicted vortex which are not confirmed in their scaled model test. Further, the strength of air entrained vortices (based on intermittency of the vortex core with time and diameter of the surface vortex) was not predicted accurately using their CFD model. Comparisons were made regarding the channel flow rate distribution for the particular pump intake geometry. Zhao & Nohmi (2012) reported discrepancy between the fixed water surface model and the VOF method used in their simulation. The authors stated that free surfaces have a significant effect on the CFD prediction of flow rate distribution in the channel flow.

Note that, even if a two-phase flow model or LES is employed, there are still some other factors that may affect CFD prediction of the formation and strength of vortices, such as mesh quality, software and its version, and boundary conditions (Matsui *et al.* 2014).

Most of the reported CFD studies on pump intakes used commercial software due to its advantages of being easy-to-learn, having a user friendly interface, and stability. The most commonly used commercial CFD software are ANSYS CFX, ANSYS FLUENT, and STAR-CD, shown in Figure 3. In-house codes were also employed but only in a few studies: for example, Chuang & Hsiao (2011) used Truchas which is developed by the Los Alamos National Lab, Roberge (1999) used FIDAP which is a general-purpose finite element CFD package, and Schneider *et al.* (2013, 2015) used a CFD package named SAM-Lattice, which is based on an LBM method.

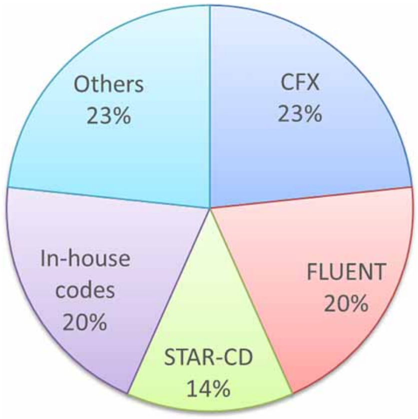

Figure 3 | Software used in CFD studies on pump intake.

IMPROVE DESIGNS AND OPERATION CONDITIONS

The ultimate goal of the previously mentioned works on CFD model development is to make CFD usable in designing a new pump intake system, troubleshooting an existing system or optimizing operation conditions. It is well known that strong vortices around the pump column or intake bell/mouth reduces pumping efficiency. The primary reasons for the formation of these vortices are poor design of the intake bay or sump (e.g. the geometric parameters that define the intake bay are the shape and size of the approach channel and pump column, the clearances between the pump column and the lateral, back and bottom walls of the channel), and uniformity level of the approach flow (Tokyay & Constantinescu 2005b). These features are also the focus of the previously reported CFD work.

Note that many fundamental principles of designing or dimensioning intake structures were developed through experiments or theoretical analysis. For example, the recommendations and formulas for dimensioning rectangular intakes, trench type intakes, suction intakes, unconfined intakes, and circular intakes are detailed per the Hydraulic Institute's standards (ANSI/HI 9.8 2012). We have to keep in mind that the use of CFD will not replace these fundamental principles but help to refine them.

Wada (2006) aimed to obtain a guideline in designing the optimum shape of the intake of vertical pumps under the condition that the installation area is minimum without losing the high pump performance. Utilizing a single-phase RANS model, he studied the expansion ratio of the intake as a representative characteristic of the shape of the pumps, and investigated the effect of the expansion ratio on pump performance. The expansion ratio is defined as D2/D1, shown in Figure 4. Wada (2006) tested expansion ratios 1.8, 1.5, 1.2, 1.1 and 1.0 and concluded that the optimum expansion ratio ranges in 1.1~1.2 to achieve a minimum installation area.

Issa *et al.* (2008, 2009) studied the impacts of the distance between pipe and end wall (i.e. X1/X2), the distance between pipe and side wall (i.e. L1/L2), submergence (S/C), and water level on vortices using a single-phase RANS model. However, no general conclusion on how to select these geometrical parameters was provided in the reports. Constantinescu & Patel (2000) simulated the flow in a water-pump intake bay and studied subsurface vortices and free surface vortices. Although they did not include design in their study, they suggested that artificial roughness on pump-bay may be used to suppress vortices and consequently increase pump efficiency.

Besides the work on a single intake system, CFD was also used to design the sump of multi-intake systems (Choi *et al.* 2010; Desmukh & Gahlot 2011; Zhan *et al.* 2012). Choi *et al.* (2010) pointed out that it is necessary to examine the flow structure around pump intake since it may reduce

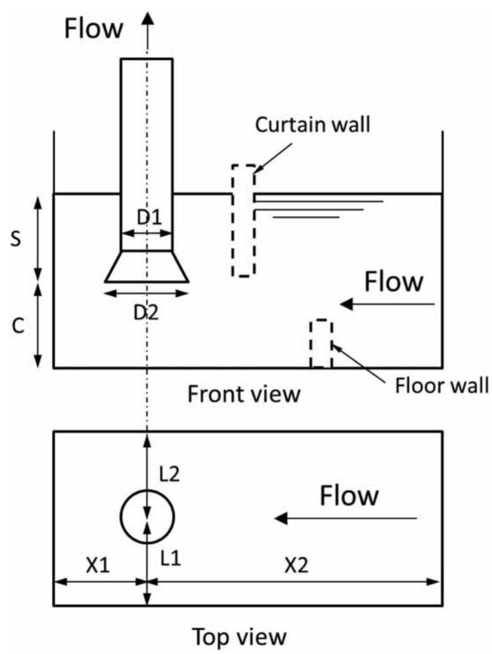

Figure 4 | Geometrical parameters in a typical pump intake system.

the capacity and efficiency of pump. They proceeded to numerically study the flows in a multi-intake pump sump model with seven pump intakes and a single-intake pump sump model. They found that one of the most important factors for the pump station design is the consideration of flow patterns at the upstream region of pump intake inlet in the forebay diffusing area. The large expanding angle at the forebay diffusing area may cause unstable fluctuating flow supply to pump intake channels and consequently increasing the intensity of vortices around the bell mouth. Desmukh & Gahlot (2011) confirmed the finding in Choi *et al.* (2010) by studying a sump model with three pump intakes (considered expanding angle 12.61° and 12.27°) and suggested reducing the expanding angle at the forebay diffusing area. Zhan *et al.* (2012) investigated five types of pump intake systems including lateral and front intakes using RANS with the VOF method. Flow uniformity was used as an index for pumping efficiency. They found that: at a relatively low flow rate, the performance of the five systems are on par with each other; for the lateral intake systems, the increase of flow rate may reduce flow velocity uniformity because of the geometrical asymmetries. Cheng & Yu (2012) also studied a lateral intake system with eight pump intakes. In order to control the inflow velocity uniformity, guide splitter was added at the inlet section of the system. The results showed that the guide splitter can significantly improve the velocity distributions at the inlet section of the sumps if installed at a proper location.

Besides modifying intake bell or sump to improve the inflow velocity uniformity, the use of a curtain wall or floor walls is another commonly used approach to eliminate or weaken vortices around the intake pipe (Tagomori & Gotoch 1989; Wicklein & Rashid 2006; Shukla & Kshirsagar 2008; Choi *et al.* 2010; Kim *et al.* 2012, 2015). The schematics of the curtain wall and floor wall are shown in Figure 4. A special combination of floor walls is also called Anti Vortices Device (AVD) (Choi *et al.* 2010; Kim *et al.* 2012, 2015). Wicklein & Rashid (2006) proposed a suite of modifications on the existing intake system, including floor walls, a curtain wall, corner fillets, and a center flow splitter, and evaluated its hydraulic performance using CFD. The results showed that the modified system can reduce the rotation of flow entering the pumps by more than one half and reduce the overall variation in the velocity distribution entering the pump by almost half. Shukla & Kshirsagar (2008) carried out two-phase flow simulations to capture air entrainment in a lab-scale pump intake with different curtain wall heights. As shown in the CFD results, air entrainment diminished or disappeared in the modified design shown in their report. Choi *et al.* (2010) found that the strong submerged vortex can be suppressed by the installation of AVD on the bottom of the pump intake channel beneath the bell mouth. Kim *et al.* (2012) also studied the impact of AVD on suppressing submerged vortex. They found that with the installation of AVD at the location of vortex emergence, the vortex disappears and the flow becomes uniform, resulting in an increase in pump efficiency by 0.9–2.1% depending on the operating flow rate. Kim *et al.* (2015) conducted further investigation on AVD using two-phase flow simulations and pointed out that the two-phase flow model improved prediction accuracy for flows around the bell mouth. This seems to be a good reason to evaluate the performance of additional wall installations using a two-phase flow model in the future.

For a pump station with multiple pump intakes, the combination of active pumps can also affect the uniformity of approach flow. Wicklein *et al.* (2006) analyzed the swirl, and the maximum and minimum velocity at the impeller location of an intake system with four pumps using CFD. Different operation modes, including 1-pump, 2-pump, 3-pump, and 4-pump modes, were considered. Based on the simulation results, a new intake system design was proposed to account for a new flow capacity, geometric changes to the influent sewer, and a new pump phase-in program. Choi *et al.* (2010) conducted CFD studies on a multi-intake pump system with seven pumps. The study compared and discussed the occurrence of the free-surface and submerged vortex under 1-pump, 3-pump, 4-pump, 5-pump, and 7-pump operation modes.

A more recent application of the CFD in pump intakes is the work conducted by Kim *et al.* (2015). Kim *et al.* (2015) studied the occurrence location of vortices and swirl angles in a multi-intake system with three pump stations using CFD. Swirl angle is the angle in which the tangential velocity forms with the axial velocity at a distance of approximately four times the diameter away from the intake. In a scaled model test, swirl angle is among the main variables used to quantitatively evaluate the design of an intake system. It is also a characteristic of swirling flow. In general, greater magnitude of the swirl angle indicates a greater tangential swirl within the intake column which is undesirable and should be avoided. The Hydraulic Institute's guideline ANSI/HI 9.8 (1998, 2012) suggests the cutoff angle to be 5°. Swirl angle greater than 5° may signal potential issues which otherwise may not be apparent to the naked eyes. Kim *et al.* (2015) conducted their work mostly to simulate the specific cause of vortices and effectiveness of AVDs but, at the same time, the work used CFD to help estimate the swirl angles for the different flow conditions. 2-pump and 3-pump operation modes were considered in their work. This article is different from the others for two reasons. The first reason is that it is one of the few CFD application for swirl angle calculations for pump intakes to address the guideline suggested by the Hydraulic Institute. Secondly, it is also one that provides quantitative measure of swirling flow inside pump intakes and correlated the CFD results with experimental results. Based on their findings, the CFD models produced results that closely resembles the trend of the experimental results, strengthening the validity and use of CFD models for pump intake simulations. The approaches considered in previous CFD simulations for improving pumping efficiency are summarized in Table 2.

SUMMARY

CFD has made significant progress towards simulating flow structures in pump intake systems. From the literature review, CFD was used in pump intake simulations for different scenarios. Various RANS and LES models were tested for their capabilities in predicting vortices in intake systems. The impacts of configuration parameters and operation conditions (e.g. the uniformity of approach flow, water depth) on pumping efficiency were also investigated using CFD simulations. CFD was also used in evaluating the performance of pump intake designs or upgrades and troubleshooting existing systems, demonstrating that it is a powerful tool. In addition, CFD simulations can reduce physical experiments significantly, making pump intake design much more efficient.

As discussed, the transient behavior of vortices and the effect of water surface are the two primary challenges in modeling flows in pump intake systems. The former can be addressed by LES while the latter can be addressed by using a two-phase flow model. However, to date, few studies have combined LES with a two-phase flow model to simulate the flow in pump intake systems. The main barrier of such a study could be the intensive computing requirements. As the computing technologies rapidly advance, there may be a better chance for resolution of this issue.

Intensive computing is also an issue to simulation of a large-scale system with a relatively complex approach channel. If feasible, it may also be advantageous to include the entire approach channel, not just a portion of it, to allow flow to be better developed before the intake.

For CFD application of multi-intake systems, a good strategy to balance accuracy and computational load could be: use the results from LES of two-phase flow to establish a relationship between velocity uniformity of approach flow and vortices occurrence and strength in a single pump intake; then use the established relationship along with the solutions of single-phase and steady flow

Table 2 | Approaches used for improving pumping efficiency and references

Improving approaches	References
Adding curtain wall/floor walls	Wicklein & Rashid (2006), Shukla & Kshirsagar (2008), Choi *et al.* (2010), Kim *et al.* (2012, 2015)
Adding guide splitter	Cheng & Yu (2012)
Sump design of multi-intakes	Choi *et al.* (2010), Desmukh & Gahlot (2011), Zhan *et al.* (2012), Cheng & Yu (2012)
Optimizing geometrical parameters of intake pipe/bell	Wada (2006), Issa *et al.* (2008, 2009)
Optimizing operation conditions (e.g. number of active pumps)	Wicklein *et al.* (2006), Choi *et al.* (2010), Kim *et al.* (2015)

simulations to further predict the vortices in multi-intake pump systems.

Besides surface vortices, the air entrainment problems may also be induced by air bubbles from non-vortex sources (Yee 2009). In addition, sediment transport and siltation at the entrance or within intakes is also important to pump intake systems (Khanarmuei *et al.* 2016). Incorporating these features into the future CFD models will allow for more comprehensive and accurate simulations of flows in pump intake systems.

ACKNOWLEDGEMENT

The authors would like to thank Christine Prouty from the University of South Florida for proofreading and the anonymous reviewers for their suggested improvements.

REFERENCES

Ansar, M. 1997 *Experimental and theoretical studies of pump-approach flow distributions at water intakes*. PhD Thesis, University of Iowa, Iowa City, IA, USA.

ANSI/HI 9.8 Hydraulic Institute Intake Design Standard 1998 The Hydraulic Institute, Parsippany, NJ, USA.

ANSI/HI 9.8 Rotodynamic Pumps for Pump Intake Design 2012 The Hydraulic Institute, Parsippany, NJ, USA.

Bates, P. D., Lane, S. N. & Ferguson, R. I. 2005 *Computational Fluid Dynamics: Applications in Environmental Hydraulics*. John Wiley & Sons, Chichester, UK.

Bayeul-Lainé, A.-C., Bois, G. & Issa, A. 2010 Numerical simulation of flow field in water-pump sump and inlet suction pipe. In: *Paper Presented at the IOP Conference Series*: *Earth and Environmental Science*.

Bayeul-Lainé, A.-C., Simonet, S., Bois, G. & Issa, A. 2012 Two-phase numerical study of the flow field formed in water pump sump: influence of air entrainment. In: *Paper Presented at the IOP Conference Series*: *Earth and Environmental Science*.

Blazek, J. 2015 *Computational Fluid Dynamics: Principles and Applications:* Butterworth-Heinemann, Oxford, UK.

Cheng, B. & Yu, Y. 2012 CFD simulation and optimization for lateral diversion and intake pumping stations. *Procedia Engineering* **28**, 122–127.

Choi, J.-W., Choi, Y.-D., Kim, C.-G. & Lee, Y.-H. 2010 Flow uniformity in a multi-intake pump sump model. *Journal of Mechanical Science and Technology* **24** (7), 1389–1400.

Chuang, W.-L. & Hsiao, S.-C. 2011 Three-dimensional numerical simulation of intake model with cross flow. *Journal of Hydrodynamics, Ser. B* **23** (3), 314–324.

Constantinescu, G. & Patel, V. 1998 Numerical model for simulation of pump-intake flow and vortices. *Journal of Hydraulic Engineering* **124** (2), 123–134.

Constantinescu, G. & Patel, V. 2000 Role of turbulence model in prediction of pump-bay vortices. *Journal of Hydraulic Engineering* **126** (5), 387–391.

Desmukh, T. S. & Gahlot, V. 2011 Numerical study of flow behaviour in a multiple intake pump sump. *International Journal of Advanced Engineering Technology* **2** (2), 118–128.

Fujii, K. 2005 Progress and future prospects of CFD in aerospace – wind tunnel and beyond. *Progress in Aerospace Sciences* **41** (6), 455–470.

Gao, Z., Jiang, C. & Lee, C. 2013 Improvement and application of wall function boundary condition for high-speed compressible flows. *Science China Technological Sciences* **56**, 2501–2515.

Issa, A., Bayeul-Lainé, A.-C. & Bois, G. 2008 Numerical Simulation of Flow Field Formed in Water Pump-Sump. In: *Paper Presented at the IAHR-24th Symposium on Hydraulic Machinery and Systems*.

Issa, A., Bayeul-Lainé, A.-C. & Bois, G. 2009 Numerical study of the influence of Geometrical Parameters on flow in water Pump-Sump. In: *Paper Presented at the Conference on Modelling Fluid Flow (CMFF'09)*.

Khanarmuei, M., Rahimzadeh, H., Kakuei, A. & Sarkardeh, H. 2016 Effect of vortex formation on sediment transport at dual pipe intakes. *Sādhanā* **41** (9), 1055–1061.

Kim, C., Choi, Y., Choi, J. & Lee, Y. 2012 A study on the effectiveness of an anti vortex device in the sump model by experiment and CFD. In: *Paper Presented at the IOP Conference Series: Earth and Environmental Science*.

Kim, C.-G., Kim, B.-H., Bang, B.-H. & Lee, Y.-H. 2015 Experimental and CFD analysis for prediction of vortex and swirl angle in the pump sump station model. In: *Paper Presented at the IOP Conference Series: Materials Science and Engineering*.

Li, S., Silva, J. M., Lai, Y., Weber, L. J. & Patel, V. 2006 Three-dimensional simulation of flows in practical water-pump intakes. *Journal of Hydroinformatics* **8** (2), 111–124.

Matsui, J., Sugino, Y. & Kawakita, K. 2014 *Numerical Simulation on Flow in Pump Sump with Free Surface*.

Okamura, T. & Kamemoto, K. 2005 *CFD Simulation of Flow in Model Pump Sumps for Detection of Vortices*. 8th Asian International Fluid Machinery Conference, Yichang, China.

Pope, S. B. 2000 *Turbulent Flows*. Cambridge University Press, Cambridge, UK.

Pradeep, S., Sayantan, G., Prasad, P. & Mohan Kumar, M. 2012 CFD simulation and experimental validation of a horizontal pump intake system. *ISH Journal of Hydraulic Engineering* **18** (3), 173–185.

Rajendran, V., Constantinescu, S. & Patel, V. 1999 Experimental validation of numerical model of flow in pump-intake bays. *Journal of Hydraulic Engineering* **125** (11), 1119–1125.

Roberge, J. A. 1999 *Use of Computational Fluid Dynamics (CFD) to Model Flow at Pump Intakes*. Worcester Polytechnic Institute, Worcester, MA, USA.

Samsudin, M., Munisamy, K. & Thangaraju, S. 2015 Application of multiphase modelling for vortex occurrence in vertical pump intake-a review. In: *Paper Presented at the IOP Conference Series: Materials Science and Engineering*.

Schneider, A., Conrad, D. & Böhle, M. 2013 Numerical simulation of the flow field in pump intakes by means of Lattice Boltzmann methods. In: *Paper Presented at the IOP Conference Series: Materials Science and Engineering*.

Schneider, A., Conrad, D. & Böhle, M. 2015 Lattice Boltzmann simulation of the flow field in pump intakes – a new approach. *Journal of Fluids Engineering* **137** (3), 031105.

Shukla, S. N. & Kshirsagar, J. 2008 Numerical prediction of air entrainment in pump intakes. In: *Paper Presented at the Proc of the 24th International Pump Users Symposium*.

Škerlavaj, A., Škerget, L., Ravnik, J. & Lipej, A. 2011 Choice of a turbulence model for pump intakes. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy* **225** (6), 764–778.

Tagomori, M. & Gotoh, M. 1989 Flow patterns and vortices in pump-sumps. In: *Paper Presented at the Proceedings of the International Symposium on Large Hydraulic Machinery*. China Press, Beijing, China.

Takata, T., Kawata, Y., Kobayashi, T., Taniguchi, N. & Morinishi, Y. 1992 Large eddy simulation of unsteady turbulent swirl flow in a pump intake. In: *Paper Presented at Computational Fluid Dynamics '92 - Proceedings of 1st European Computational Fluid Dynamics Conference, Brussels, Belgium, 7–11 September 1992*, pp. 255–261.

Tejada-Martinez, A. E. & Grosch, C. E. 2007 Langmuir turbulence in shallow water. Part 2. Large-eddy simulation. *Journal of Fluid Mechanics* **576**, 63–108.

Tokyay, T. & Constantinescu, S. 2005a *Investigation of Flow Physics of Pump Intake Flows Using Large Eddy Simulation.* University of Iowa, Iowa City, IA, USA.

Tokyay, T. & Constantinescu, S. 2005b Large eddy simulation and Reynolds averaged Navier-Stokes simulations of flow in a realistic pump intake: a validation study. In: *Impacts of Global Climate Change*, pp. 1–12.

Tokyay, T. & Constantinescu, S. 2006 Validation of a large-eddy simulation model to simulate flow in pump intakes of realistic geometry. *Journal of Hydraulic Engineering* **132** (12), 1303–1315.

Wada, A. 2006 Flow structure around the intake of a vertical pump. *Journal of Thermal Science* **15** (2), 121–125.

Wicklein, E. & Rashid, M. 2006 Use of Computation Fluid Dynamic Modeling to Evaluate Pump Intake Performance and Develop Design Modifications. In: *Paper Presented at the World Environmental and Water Resource Congress 2006: Examining the Confluence of Environmental and Water Concerns*.

Wicklein, E., Sweeney, C., Senon, C., Hattersley, D., Schultz, B. & Naef, R. 2006 Computation fluid dynamic modeling of a proposed influent pump station. *Proceedings of the Water Environment Federation* **2006** (5), 7094–7114.

Wu, Y., Li, Y. & Li, X. 2000 PIV experiments on flow in a model pump suction sump. *Research Report*, Tsinghua University, Beijing, China.

Yee, T. M. 2009 *Three-Dimensional Free Surface Non-Hydrostatic Modeling of Plunging Water with Turbulence and Air Entrained Transport*. Doctoral dissertation, University of Kentucky, Lexington, KY, USA.

Zhan, J., Wang, B., Yu, L., Li, Y. & Ling, T. 2012 Numerical investigation of flow patterns in different pump intake systems. *Journal of Hydrodynamics, Ser. B* **24** (6), 873–882.

Zhang, J., Tejada-Martínez, A. E. & Zhang, Q. 2014 Developments in computational fluid dynamics-based modeling for disinfection technologies over the last two decades: a review. *Environmental Modelling & Software* **58**, 71–85.

Zhao, L. & Nohmi, M. 2012 Numerical simulation of free water surface in pump intake. In: *Paper Presented at the IOP Conference Series: Earth and Environmental Science*.

First received 22 August 2017; accepted in revised form 21 December 2017. Available online 4 January 2018