Extensive Journal of Applied Sciences
Available online at www.ejasj.com ©2025 EJAS Journal-2025-09-30, 9-23

Application of Modern Technique to Predict and Enhance Flow Problems on Pump Suction Intake for Irrigation Improvement Projects

Ibrahim R. Teaima

National Water Research Center, Mechanical and Electrical Research Institute, Egypt

Corresponding author email: Ibrahim R.Teaima

ABSTRACT: The Irrigation Improvement Project (IIP) is being implemented at a national scale in Egypt with a target of improving the old irrigation system in the Nile Delta for increasing water use efficiency and agricultural production. The objective of this research is to study the influence of existing circular sump shape on the flow conditions of mesqa pump station. The adverse flow conditions at a pump intake lead to occurrence of swirl and vortices, which in turn reduce the pump efficiency, induce vibrations and excessive bearing loads resulting in operational and maintenance cost and leading to other operating difficulties. The field measurements are conducted on the Efrank (1) mesqa pumping station at Kafr El-Dawar command area, El-Beheara Governorate. The 3D CFD model solves the Reynolds averaged Navier–Stokes (RANS) equations with the k–ε turbulence model with wall function for predicting the occurrence of vortices and swirl in pump intakes. Various modification of existing circular sump shapes are studied, in both circular and rectangular shaped sumps. The modifications are tested under the wide range of sump water level. These modifications are responsible to reduce or eliminate the flow pre-swirling and vortex. In this study, three different modifications are done, installing submerged cone under the suction pipes, installing vertical splitter wall between two suction pipes and using rectangular sump shape. The research shows that, existing circular sump shape generates swirl flow and vortex at the sump and suction pipes of pumping units. The submerged cone under suction pipes doesn't reduce the swirl flow and vortex. Furthermore, when using rectangular sump shape, the stream line plots show occurrence of two strong vortices on the water surface. The installation of vertical splitter wall between the two vertical suction pipes reduces the pre-swirling surface flow and improves the pump performance.

***Keywords*:** Irrigation improvement; vortices; pre-swirling; pump sump; flow problems.

INTRODUCTION

It is an accepted fact that faulty design of pump sump or intake is one of the major causes of unsatisfactory operation of pumps in any pumping station. The efficiency and performance of pumping stations involving multiple pumping units depend not only on the efficiency of the pumping units but also on the proper design of the intake sump. Recently, smaller size suction pump sumps are required to reduce the construction cost of mesqa pumping stations in Egypt. To design such compact and high-speed pump sumps, it is necessary to avoid the generation of strong vortices that induce harmful vibration and noise. According to the JSME criteria for a pump sump design [1], a submerged vortex with a vapor core should be precluded. So far, experiments on scale models had been

performed to assess sump performance and to develop a suitable sump design. These experiments, however, are costly and time-consuming. Hence, alternative evaluating methods are needed. Kamemoto et al. [2] presented numerical results of basic flow features in a pump sump by using a three-dimensional discrete vortex method. Takata et al. [3] reproduced a strong cavitating vortex in a pump sump by the Large Eddy Simulation. More practical codes based on the Reynolds Averaged Navier Stokes (RANS) equations were also applied to flow in pump sumps, and it was shown that general flow patterns, which induce the submerged vortices, agreed well with experiments [4-5]. The common solution for avoiding air-entrainment and swirl is to provide sufficient submergence to the intake. If the required approach flow conditions cannot be met to avoid swirl and air entrainment, it is economical to consider other approaches for preventing vortices at water intakes. While the effect of extreme pressure variation in a pressurized flow due to air entrainment is shown by Kabiri-Samani et al. [6]. There are several means of avoiding air-entrainment where for large structures; the most cost-effective option is often determined by a physical model study. One of the most popular ways is the use of special vortex suppression devices [7]. Ample research work has been carried out to have a better knowledge of vortex formation and critical submergence depths for vertical pipe intakes. Among the analytical approaches, the works of Odgaard [8] and Hite and Mih [9] are more relevant. However, for experimental published work, the studies can be divided into two categories. First, the understanding of free vortex hydrodynamics [10-15] and second, parameters affecting the critical submergence depth [11,16-20]. On the other hand, while many attempts have been made numerically to analyze the hydrodynamics of vortex and cavity development [21, 22], not much published work with a numerical basis exists on air entrainment due to shaft intake.

The objective of this research is to study the modifications of existing circular sump intake of irrigation improvement projects in mesqa pumping stations of the improved mesqas, in future IIP projects instead of using circular sump. This change in the design of the improved mesqas, is an attempt to lower the cost of mesqa operation and maintenance costs per feddan, which will be recovered from the farmers due to increase the efficiency of pumping units.

Describing the Study Mesqa

The Irrigation Improvement Project (IIP) is a project which is implemented in order to increase water use efficiency and agricultural productivity in Egypt's old lands [23]. The Efrank (1) mesqa pumping station located at Km 3.4 on the right-hand side of El- El-Baslaqon Canal. It serves a total command area of about 48.5 feddan. The pipeline material is composed of PVC with diameter of 200 mm and 670 m length. The pipeline has nine outlets at intervals along its length which serves quaternary units (Marwas). Each outlet has a butterfly valve allowing water to be discharged into open channel. The pumping station consists of two small single stage end suction centrifugal pumps, flow rate 30 L/s, head 13 m, rated horse power 10 at 1450 rpm. The sump of Efrank (1) mesqa pumping station is circular shape with 170 cm diameter and 300 cm depth made from concrete. The sump consists of two vertical steel suction pipe lines with nominal diameter of 5 inch connected with pump suction sides with elbow and flexible joints as shown in Figure (1).

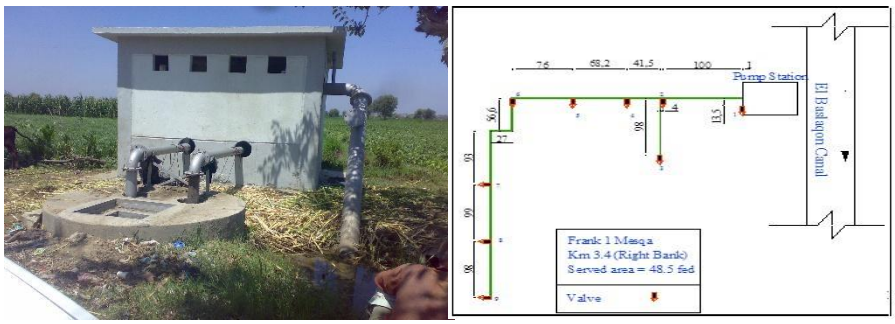


Figure 1. Efrank (1) mesqa pumping station with circular sump

Executing Field Measurements

The hydraulic performance evaluation of Efrank (1) mesqa pumping station is done in two units. After long period of operation the concrete base of pumping units is exposed to breakdown and crack due to the excess vibration of pumping unit's especially at low water level at El-Baslaqon Canal. A transit-time ultrasonic flow-meter type (Siemens) is used to measure the volume flow rate through the pipeline. Calibrated pressure transducers are used to measure pressures at the suction and delivery sides of the pumping units. Three phase energy analyzer, (Fluke 434) is used to measure voltage, ampere, active power, energy, apparent power, frequency and power factor. The pressure head developed by the pump is recorded with time at different operating conditions.

Executing Numerical Studies

From a purely numerical perspective the geometric complexity of the problem is such that it demands the full power of modern computational fluid dynamics (CFD) to solve the equations of motion and turbulence models in domains that involve multiple surfaces. Additional difficulties are associated with modeling free surface and vortex phenomenon, the physics of which is not yet fully understood. In spite of the practical importance of the problem, the literature on numerical modeling of pump intake flows is rather limited. In the present work an attempt has been made to simulate and predict the flow conditions such as vortices and swirl for multiple sump shapes, with an aim to determine the viability of, commercially available computational fluid dynamic software (Ansys 14.5 Fluent) as an important design optimization tool for intake sumps.

CFD model

This model solves the Reynolds-averaged Navier–Stokes (RANS) equations for steady, incompressible turbulent flows in a pump intake. The mass and momentum equations are written, in Cartesian tensor form, as

$$\frac{\partial U_j}{\partial x^j} = 0 \quad (1)$$

$$U^t i \qquad U^x i^j U j \qquad {}^x P^i \qquad {}^{x\,j} \qquad U^{x\,j} i \qquad uiu\,j \quad (2)$$

where U_j and u_j are jth components of the mean and fluctuating velocities, P is the piezometric pressure and ρ and μ are fluid density and viscosity, respectively.

The Reynolds stresses viscosity formula: $u^i u^j$ are expressed in terms of the mean rate of strain using Boussinesq's eddy-

$$u^i u^j \quad {}^t \quad Ux\,i \; UU\,j \quad 2\;k^{\,ij} \quad (3)$$

where μ_t = turbulence viscosity, δ_{ij}= the Kronecker symbol and k= turbulence kinetic energy per unit mass. For the present applications, the standard k– ε model [24] is used. The eddy viscosity is obtained from $k2$

$${}^t \quad C \quad (4)$$

where ε= turbulence dissipation rate. The transport equations for k and ε are expressed as

$$\frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right) + G - \rho\varepsilon \quad (5)$$

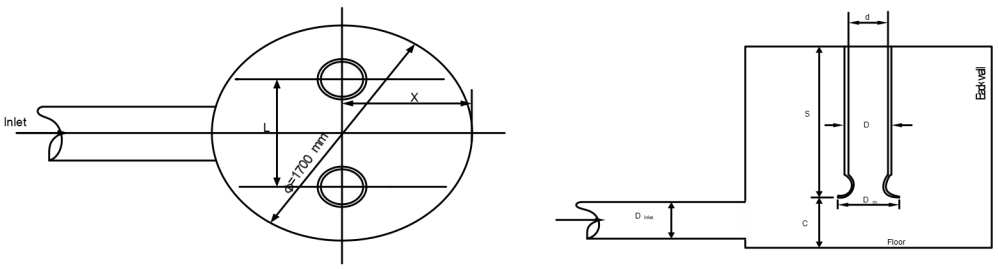
$$\frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j}\left(\Gamma_\varepsilon \frac{\partial \varepsilon}{\partial x_j}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}G - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \quad (6)$$

Where $G = -\rho\overline{u_i u_j}\frac{\partial u_j}{\partial x_i}$ is the generation of k. The tur bulence model coefficients are: C_μ=0.09, σ_k= 1.0, σ_ε= 1.3, $C_{\varepsilon 1}$=1.44 and $C_{\varepsilon 2}$= 1.92.

The finite volume approach is employed for the discretization of the governing equations. The discretization in space is of second order accuracy. The physical domain is meshed into a number of cells in which all dependent variables are stored at the geometric centers of each volume cell. The equations are integrated over cells using the Gauss theorem.

Description of the intake model and numerical simulation

Figure (2) shows the overall pump intake configuration of the pump intake model simulated in this study. Figure (2-a) provides a plan view of the full domain and Figure (2-b) reflects a vertical view of a longitudinal cross section passing through the pipe center. In the figure, d and D are the inner and outer pipe diameters, respectively, ϕ is circular sump diameter, L is the distance between two inlet sections to pipe center, X is the distance from backwall to pipe center, S is the pipe entry depth, C is the clearance from the floor and D_m is the pump bell mouth diameter at the entrance. The geometric parameters are: d=0.128 m, D=0.136 m, D_m=0.208m, Φ=1.7 m, L=6.64 d, X=3.32d, C=3.9d, H=15.62d (H is the total pipe simulated length). There are three different pipe submergence distance, S=3.9d, 7.8d and 11.7d respectively. The model discharge is Q_m =0.03 m^3/s and the Reynolds, Froude and Weber numbers based on d are 298240, 0.49 and 531.75, respectively.



(a) Plane view (b) Elevation

Figure 2. Configuration of the Efrank (1) meska pumping station intake.

Geometry and grid generation

The basic three-dimensional geometry is prepared using FLUENT's preprocessor GAMBIT 2.2.30 drawing with the grid surfaces plot of sump. Three dimensional unstructured mesh is used for the flow simulation in the pump sump. Unstructured mesh is used for this study due to model complexity and easy to be mesh especially at the intake section. The numerical solver uses unstructured meshes which allow flexibility in meshing very complex geometries while maintaining high quality computational mesh which is necessary for obtaining accurate solutions [25]. The three-dimensional unstructured mesh (tetrahedral) has been created by using the structured mesh (hexagonal) as shown in Figure (3-a). Total elements for this model are 965,270 with the maximum aspect ratio 16.63. After the meshing is completed, the boundary condition was set through a command zone provided by pre-processor. The 'velocity inlet' was set at inlet while 'pressure outlet' for the outlet region. Water surface are set as 'symmetry' boundary and pump model is set as 'walls' including the bell moth as shown in Figure (3b).

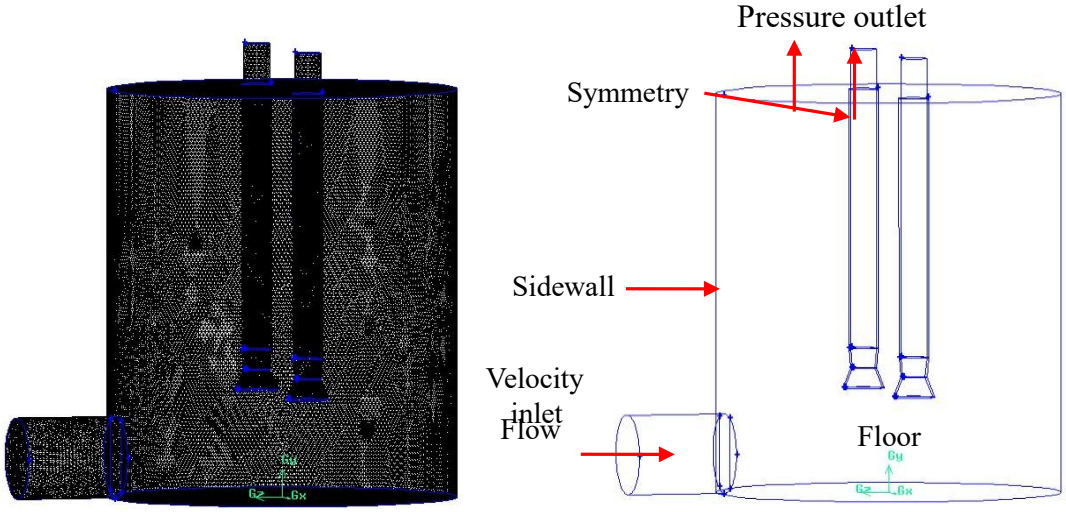


Figure 3. Efrank (1) mesqa pumping station geometry, (a) computational grid (b) boundary condition setup.

RESULTS AND DISCUSSION

Field measurements

Hydraulic performance tests are done during normal operating conditions around the design point, where the control valve in the delivery pipe is used to control flow rate. Discharge, suction pressure, delivery pressure, power quality values are monitored and total head is computed for each unit. Figure (4) shows the hydraulic test performance when the two pumps operate at the same time and the water suction level depth, S=11.71 d. Figure (4) shows the manufacture (H-Q) curve of the pump at different operation conditions. From the figure, the actual operating points of each pump measured at high water level in the sump are closely with the manufacture pump head. Furthermore, the measured flow rate higher than the design values is due to decrease of the total head produced by each pump. Figure (5) shows also the operating point of each pump at the suction water level at sump reach to, S=7.8 d. From this figure it can be illustrated that, the measured values of flow rate and total head don't match the manufacturing (H-Q) curve but the pumps deliver the required discharge. Consequently, when decreasing the suction water level at the circular sump of meska as shown in Figure (6) reach to, S=3.9 d the pump doesn't match the operation point according to ISO 9906. It is believed that, the big deviation between the measured performance values and the standard manufacturing curves is due to strong vortex and swirling at the suction circular sump, consequently the pumps are exposed to hydraulic problems in the pump suction such as the cavitation phenomena, which leads to pump

vibrations, impeller damage, wear due to cavitations, deteriorated performance and a significant increase in operational and maintenance costs.

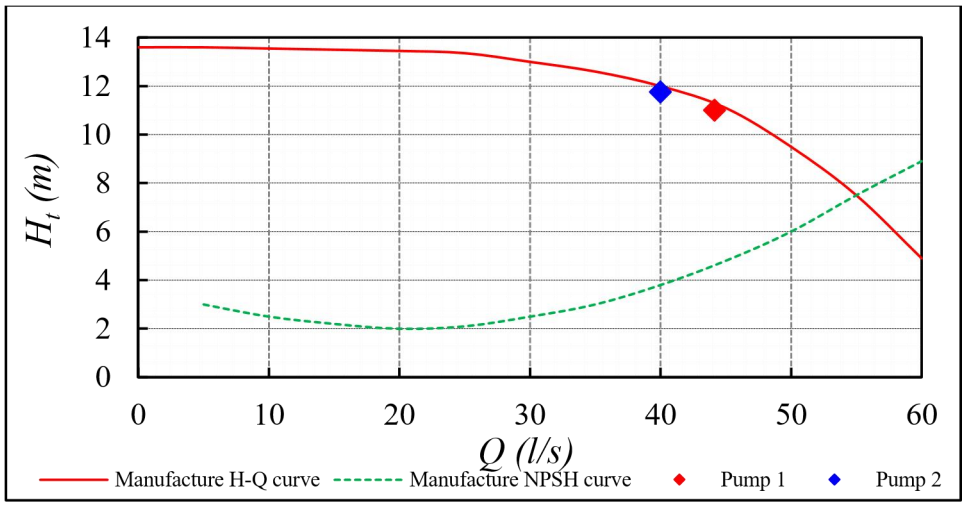


Figure 4. Characteristics curves at high suction water level, S=11.71 d.

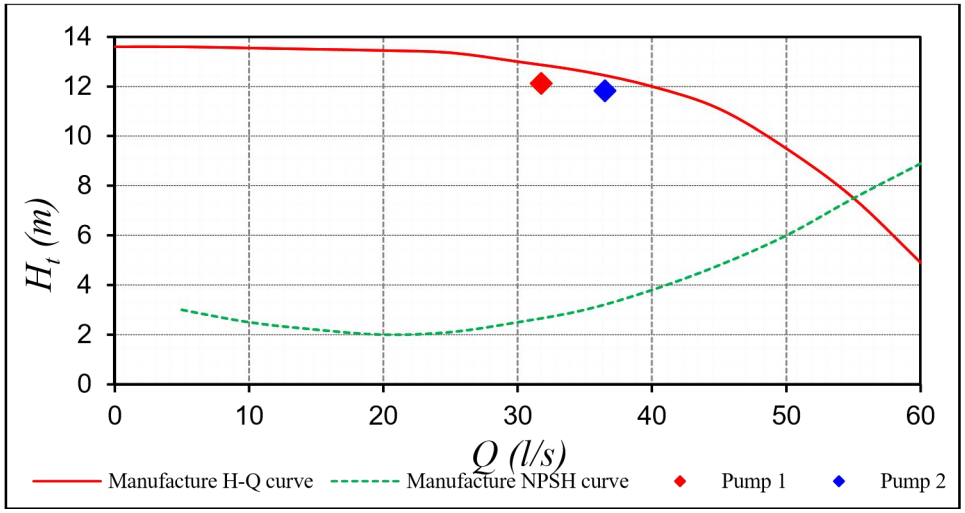


Figure 5. Characteristics curves at medium suction water level, S=7.8 d.

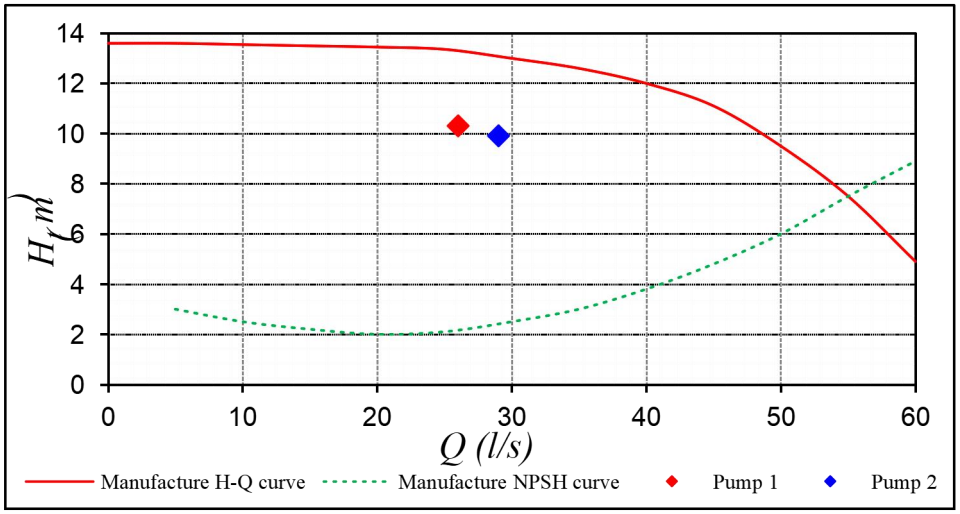
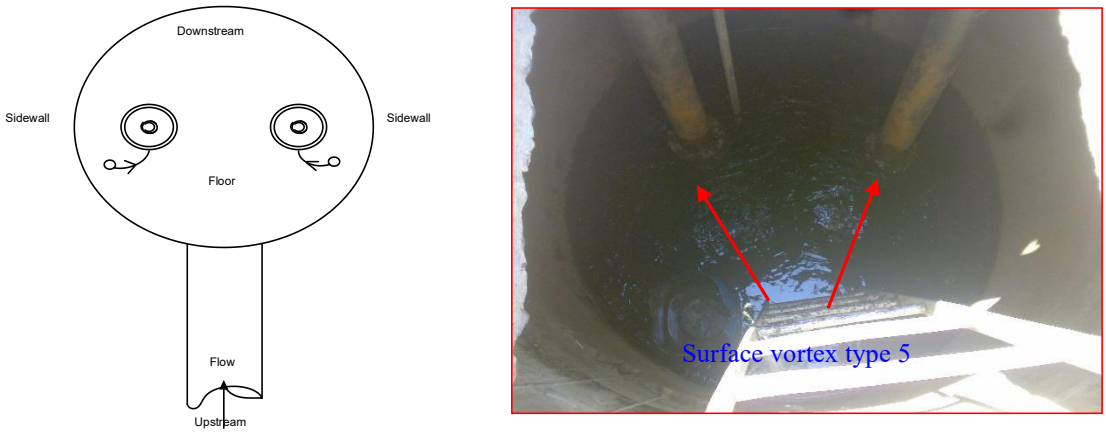


Figure 6. Characteristics curves at low suction water level, S=3.9 d.

Visual observations

Figure (7-a) shows the formation of vortices in the sump. Figure (7-b) shows the visual observation of vortex and swirling flow when decreasing the suction water level at circular sump to S=3.9 d above

the sump floor. The initial location of the vortex is shown by the “o” and the line illustrates the typical path of the vortex as shown in Figure (7-b). The visual observation of the vortices formed at both sides of the intake reveals the symmetrical position of the vortices. The vortex Type 5 is observed consistently with the addition help of dye tracing technique [8]. Further, the vortex Type 5 is pulling air bubbles into the suction intake, for a long period of time. The occurrence of this type of vortex are forming and breaking more than 30s. These could permanently damage the impeller of the pump. This problem will lead to the decreasing of pump performance and efficiency. It also could cause high maintenance and operational costs [26]. The positions of subsurface vortex are directly under the suction intake (bell mouth). This subsurface vortex is formed in 10s to 15s cycles as a fine threadlike line [27]. This action will further affect the optimum of pump performance because it reduces pressure in the vortex core that can possibility increase the cavitations. The submerged vortex cavitation interacts with the pump impeller affecting the pump’s reliability. Furthermore, strong vortex causes vibration and noise in the pumps operation
[28].



(a) (b)

Figure 7. (a) The Formation of vortices exists in physical experiment (b) Surface vortex and Sub-surface vortex

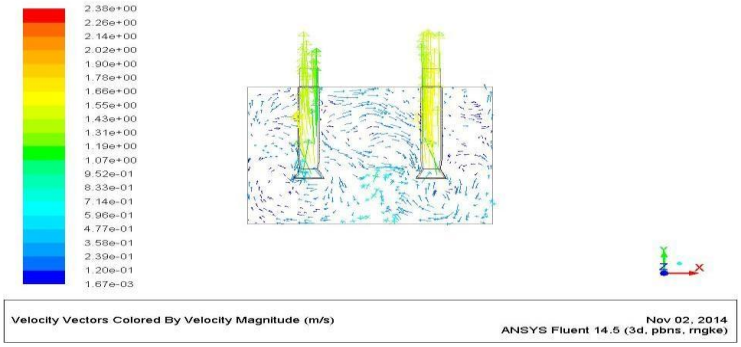
Numerical results

Different approaches are simulated using CFD in the following studies to find the optimum and reliable modification method to increase efficiency of the Irrigation Improvement Projects (IIP) stations. This section divided into two parts. The first part contains the numerical study of the original circular sump with diameter of 1.7 m and the total sump depth is 3 m when two pumps are in operation. The second part contains the numerical studies of modification sump. This modification is subjected to small sump modifications such as put cone under suction pipe intake or put the wall splitter between the two vertical suction pipes. Furthermore the major sump modifications represent the construction of the rectangular sump shape.

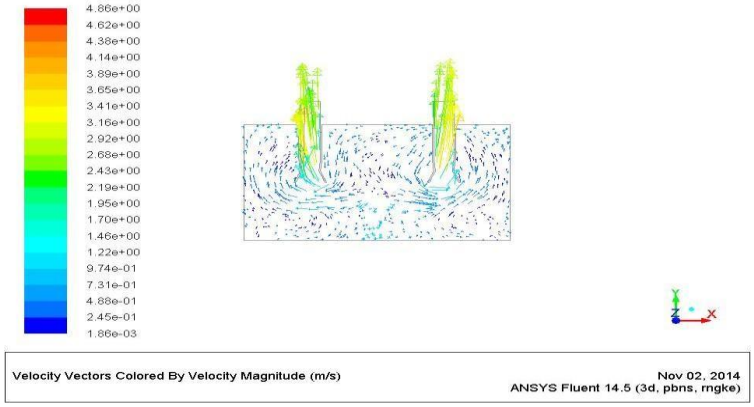
Original circular sump model

The original model consists of circular sump with internal diameter of 1.7 m, the total distance between the two vertical suction pipes is 850 mm, the distance between the suction pipe and back wall equals 425 mm, the distance between the sump floor and intake of suction pipe is 500 mm. In this study there are three different water levels in the sump, S=3.9, 7.8, 11.71 d respectively.

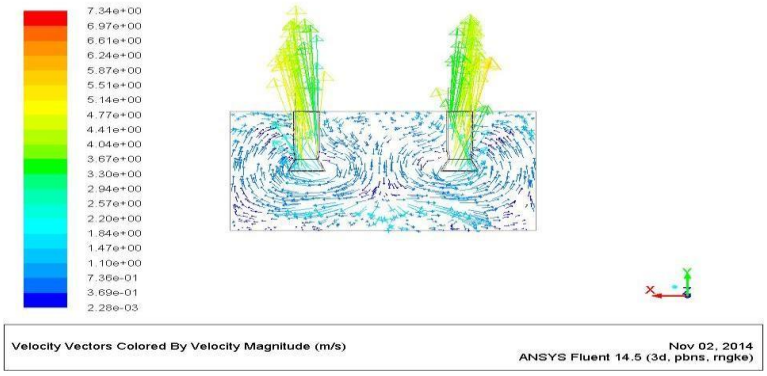
bFigure (8) shows the velocity vector distribution in circular sump at different three water levels S. From this figure it can be illustrated that, the velocity distribution at the low water level, S=3.9 d, the circulation separation zones appeared at the sump intake under the suction pipe and behind the sump floor.



(a) S=11.7d



(b) S=7.8d



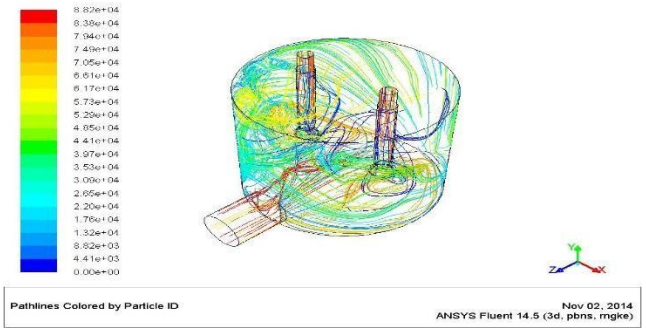
(c) S=3.9d

Figure 8. Velocity vectors distribution at different water level.

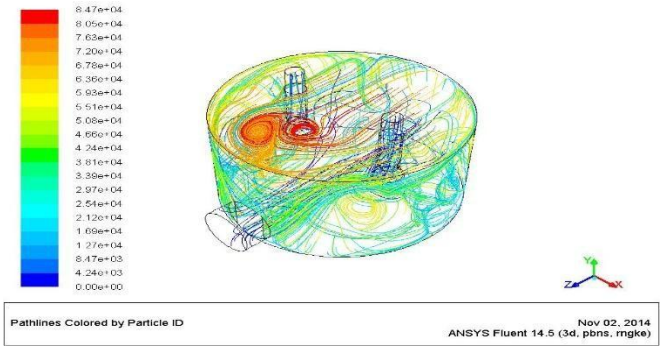
The streamline patterns in the original circular sump of improvement projects are shown in Figure (9). As illustrated in Figure (9-a), when the suction water level, S=11.7 d, free surface vortices are merged into a strong vortex under the bell mouth and discharged with a high velocity. Relatively weak subsurface vortices of Type 5 [1] also occur on the both side walls but still no pulling air and no cavitation are shown in this case. In the case of the S=7.8 d, as shown in Figure (9-b), bell mouth, weak free surface vortices are appeared but the subsurface vortices at both side wall and the vortex strength in the intake pipe from the bell mouth are more complicated and strong. Consequently when the water level decreases to reach minimum values, S=3.9 d, as shown in Figure (9-
c), a very large rotating mass of fluid is created in the free surface of circular sump. Very strong vortices are appeared in the free surface and subsurface. Consequently, the existing circular sump shape generates swirl flow and vortex at the sump and suction pipes of pumping units. A whole aspect of vortex flow behavior is well understood in this figure. The above observations show the presence of undesirable flow conditions and poor sump design. The problems associated with the poor sump

design are vibrations, impeller damage / wear due to cavitations, deteriorated performance and a significant increase in operational and maintenance costs.

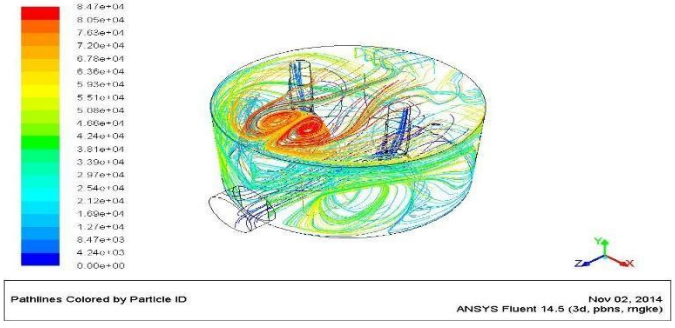
Improvement in the flow conditions can be achieved through suitable modifications in sump geometry and/or providing appurtenances. In the present case, to minimize these disturbances, a number of variants of the original sump model with modifications in different elements of the sump geometry such as put the cone under each vertical pipe intake, put the splitter wall between the two vertical pipes and using the rectangular sump shape, are tried out and the numerical simulation for each of these modifications is performed.



(a) S=11.7d



(b) S=7.8d



(c) S=3.9d

Figure 9. Streamlines and vortex isosurface plot in circular sump at different water level.

Modification of testing sump

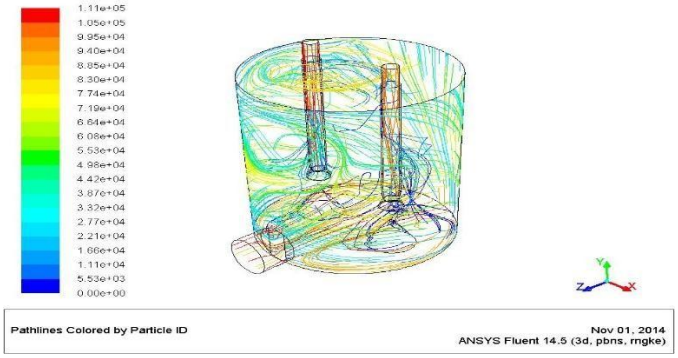
Following the original circular sump design testing, a series of design modification tests is conducted. The objective of these tests is to develop and implement changes to the original design that would reduce or eliminate the adverse flow conditions observed for the circular sump geometry. Modification is conducted at the specified three different water levels in the sump. A total of three different geometrical arrangements are investigated in the model as part of modification. Modifications examined in the model are included the following:

- Installation cone wall, under the center line of each vertical suction pipe and fixed at circular sump floor;

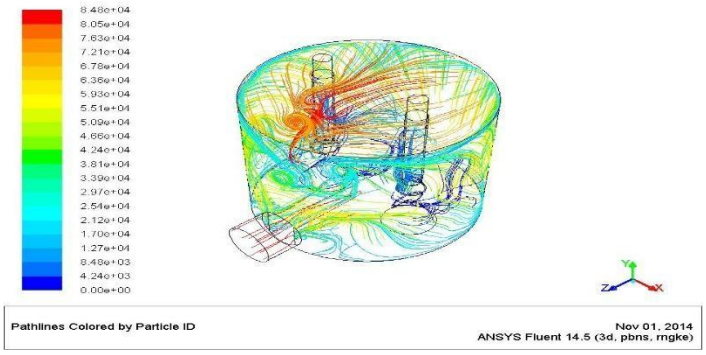
• Installation of a splitter wall between two vertical suction pipe and fixed from circular sump floor to the water free surface;

• Using the rectangular sump shape with the same circular sump size;

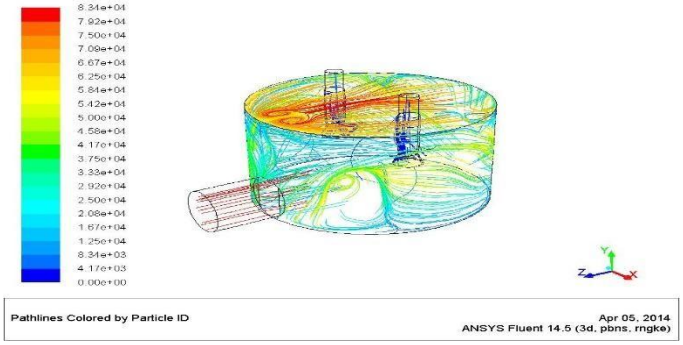
Figure (10) shows the installation of cone wall scenario. The cone wall is often utilized to reduce or eliminate sub-surface vortex activity along the boundaries of the sump in the vicinity of the pump bell mouth. The subsurface swirling flow is appeared when suction water level at the circular sump intake with submerged cone reaches to, S=11.71 d as shown in Figure (10-a). Also the free surface vortex and swirling flow occur when the suction water level reaches, S=7.8 d as shown in Figure (10-b). Consequently, at low suction water level at the circular sump with submerged cone, S=3.9 d, the twin free surface vortex and sub-surface vortex are illustrated as shown in Figure (10-c). Based on these results, the installation of submerged cones within the circular sump pump is not recommended because this cone not decreasing the swirl flow and vortex and can generate the hydraulic and mechanical problems in the pumping stations with circular sump.



(a) S=11.7d



(b) S=7.8d



(c) S=3.9d

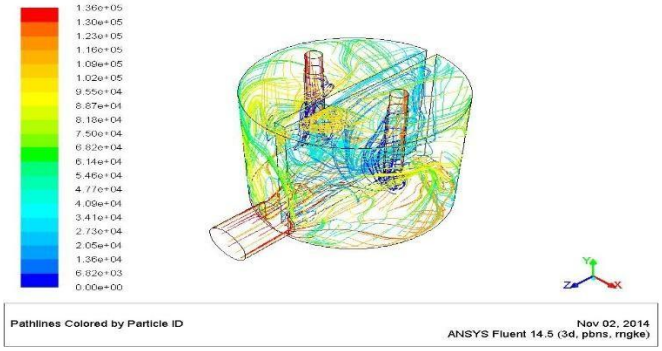
Figure 10. Streamlines and vortex isosurface plot in circular sump when using cone wall at different water level.

Figure (11) illustrates the installation of splitter vertical wall between the two vertical suction pipes. Pump sump designs frequently examine the use of a splitter wall as a method of reducing or eliminating the level of surface vortex activity, velocity non-uniformity, and flow pre-swirl. At this

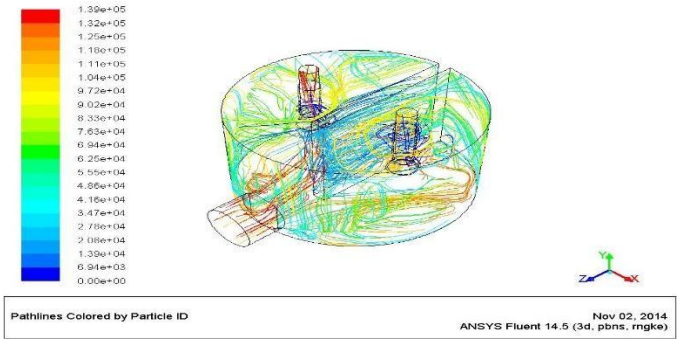
stage of the test program as shown in Figure (11-a), no significant surface vortex activity in the vicinity of the circulating water pumps had been observed. Hence, testing is conducted to examine the use of a splitter vertical wall as a method of reducing or eliminating velocity non-uniformity and flow pre-swirl. For the present study, testing is conducted with a vertical wall installed in region between the two suction pipes with thickness of 10 cm and vertical height between the circular sump floor and water free surface. It is found that the placement of the splitter vertical wall is influential on reducing velocity non-uniformity and the intensity of flow pre-swirl as shown in Figure (11-b). For example, by setting the water level at S=3.9 d, the flow pre-swirl is reducing compared to in the case of original circular sump and when using the cone under the suction vertical pipe as shown in Figure (11-c). Although the splitter wall is effective in reducing flow pre-swirl angle and velocity non-uniformity, its installation is still optional given the fact that the sump meets all performance criteria with just the installation of the separate vertical wall as described above.



(a) S=11.7d



(b) S=7.8d

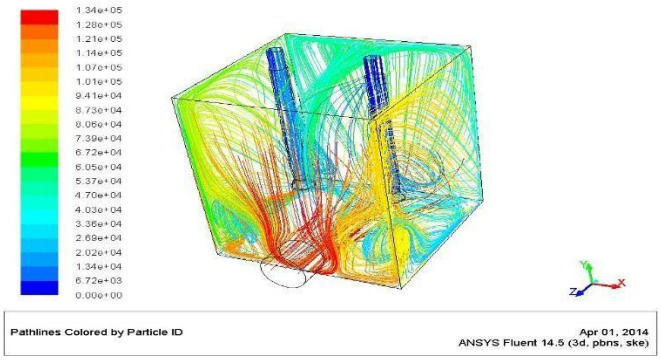


(c) S=3.9d

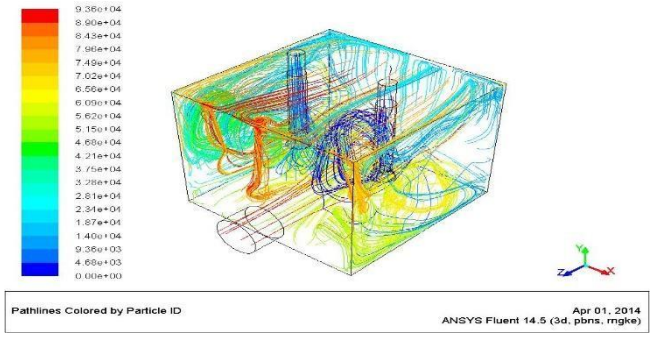
Figure 11. Streamlines and vortex isosurface plot in circular sump when using vertical splitter wall at different water level.

Figure (12) shows the installation and use the rectangular sump. Rectangular sump is often used at the most pumping stations as an alternative method of reducing flow pre-swirl and velocity non-uniformity. The rectangular sump installation with dimension of 1.7 m depth and 2 m width with 3 m

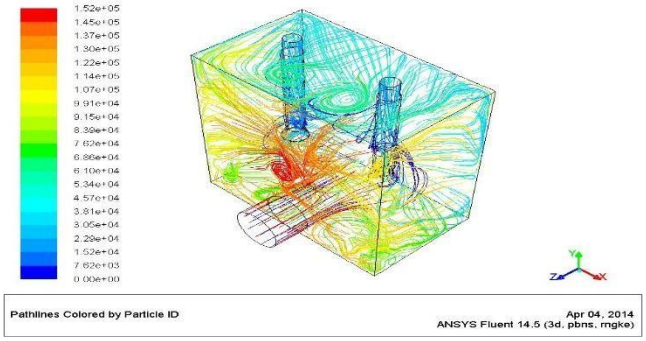
height, are examined numerically at different three water levels at the sump, S=3.9, 7.8 and 11.7 d respectively. The streamline plots show occurrence of two strong vortices on the water surface as shown in Figure (12-b). Consequently when decreasing sump water level to S=3.9 d, another submerged vortex appears at sub surface at front side of the vertical suction pipes as shown in Figure (12-c). Hence the suitability of using rectangular sump configuration is not confirmed even for total flow rate of 0.06 m^3/s.



(a) S=11.71 d



(b) S=7.8 d



(c) S=3.9 d

Figure 12. Streamlines and vortex isosurface plot in rectangular sump at different water level.

Nomenclature

C clearance from floor to pipe entry
$C_{1\varepsilon}$, $C_{2\varepsilon}$ constants in the turbulence transport equations, k–ε model
C_μ Kolmogorov constant d
inner pipe diameter D
outer pipe diameter
D_m pump bell mouth diameter at the entrance
D_{inlet} inlet diameter of pipe intake g
acceleration due to gravity
G generation of turbulence kinetic energy
H total pipe length

H_t pump monomeric head k turbulence kinetic energy

L distance between two vertical suction pipes

P piezometric pressure

Q_m model discharge

Q pump discharge S pipe entry depth

u_i fluctuating velocity components (i=1,2,3)

U_i mean velocity components (i=1,2,3)

$\overline{u_i u_j}$ Reynolds stresses x,y,z streamwise, transverse, and vertical direction coordinates X distance from backwall to pipe center

Greek symbols

δij Kronecker symbol (δij =1 when i= j, δij=0 when i ≠ j)

ε turbulence dissipation rate

μ_t turbulence viscosity (^{t})

t eddy viscosity

ρ water density

$\sigma k, \sigma_\varepsilon$ constants in the turbulence transport equations, k–ε model

Φ circular sump diameter

CONCLUSION

In this paper, a study on the circular sump of irrigation improvement project effects on the performance of mesqa pumping stations. The paper discusses the different modifications on pump sump, such as the installation of submerged cone under each vertical suction pipe, the installation of splitter wall between two vertical suction pipe and when using the rectangular sump shape. These modifications in sump intake are done at three different water levels in sump intake. The existing of circular sump shape generates swirl flow and vortex at the sump and suction pipes of pumping units. The conclusion of installing the vertical splitter wall between the two vertical suction pipes is reducing the pre-swirling surface flow and improving the pump performance. Further, for a rectangular sump shape, the streamline plots show occurrence of two strong vortices on the water surface. Furthermore another submerged vortex appears at sub-surface at front side of the vertical suction pipes.

Consequently the suitability of using rectangular sump configuration is not confirmed even for a flow Q_m= 0.06 m^3/s. The paper recommends also not using the submerged cone under the suction pipe, because this cone not decreasing the swirl flow and vortex and can generate the hydraulic and mechanical problems in the pumping stations with circular sump.

REFERENCES

"Standard Method for Model Testing the Performance of a Pump Sump", JSME S 004, The Japan Society of Mechanical Engineers, 1984, in Japanese. [1]

Kamemoto, K., Toyokura, T. and Sano, A., "A Numerical Simulation Technique for Predicting of Flow Feature of a Pump Sump", Proc. IAHR Symp., Montreal, Vol.2, 52, pp.1-14, 1986. [2]

Takata, T., Kawata, Y., Kobayashi, T., Taniguchi, N. and Morinishi, Y., "Large Eddy Simulation of Unsteady Turbulent Swirl Flow in Pump Intake", Proc. of Jpn. Soc. Mech. Eng., No.920-78, pp.524-

526, 1992, (in Japanese) [3] Enomoto, T., Tagomori, M. and Goto, A., “Turbomachinery”, Vol.23, pp.165-168. 1995. [4]

Constantinescu, G. S. and Patel, V. C., “Role of Turbulence Model in Prediction of Pump Intake Vortices, ASME FEDSM98-5003, 1998. [5]

Kabiri-Samani, A.R., Borghei, S.M. and Saidi, M.H., “Entrapped air in long water tunnels during transition from pressurized to free surface flow regime", Scientia Iranica, 13(2), pp. 174-186, 2006. [6]

Knauss, J., “Swirling flow problems at intakes", Hydraulic Structures Design Manual 1. A.A., IAH, Balkema, Rotterdam, 1987. [7]

Odgaard, A.J., “Free-surface air-core vortex", Journal of Hydraulics Engineering, ASCE, 112(7), pp. 610-620, 1986. [8]

Hite, E.J. Jr., Mih, W.C., ”Velocity of air-core vortices at hydraulic intakes", Journal of Hydraulics Engineering, ASCE, 120(3), pp. 284-297, 1994. [9]

Daggett, L.L. and Keulegan, G.H.,”Similitude in free surface vortex formations", Journal of Hydraulics Division, ASCE, 100(11), pp. 15651580, 1974. [10]

Anwar, H.O., Weller, J. and Amphlet, M.B., “Similarity of free-vortex at horizontal intake", Journal of Hydraulic Research, 16(2), pp. 95-105, 1978. [11]

Jain, A.K., Ranga Raju, K.G. and Garde, R.J., “Vortex formation at vertical pipe intakes", Journal of Hydraulics Division, ASCE, 104(10), pp. 1429-1445, 1978. [12]

Padmanabhan, M. and Hecker, G.E. ,”Scale effect in pump sump models", Journal of Hydraulics Division, ASCE, 110(10), pp. 1540-1556, 1984. [13]

Rocklage-Marliani, G., Schmidts, M. and Ram, V.I.V.,”Three-dimensional laser-Doppler velocimeter measurements in swirling turbulent pipe flow", Applied Scientific Research, Springer Netherlands, 70(1-4), pp.43-67, 2003. [14]

Carriveau, R.,”The hydraulic vortex - an autocatakinetic system", Int. Journal of General Systems, 35(6), pp. 707-726, 2006. [15]

Reddy, Y.R. and Pickford, J.A., “Vortices at intake in conventional sumps", Water Power, No. 3, pp. 108-109, 1972. [16]

Yildirim, N. and Kocabas, F., ”Critical submergence for intakes in open channel flow", Journal of Hydraulics Division, ASCE, 121(12), pp. 900-905, 1995. [17]

Yildirim, N. and Kocabas, F., ”Prediction of critical submergence for an intake pipe", Journal of Hydraulic Research, IAHR, 40(4), pp. 507518, 2002. [18]

Borghei, S.M. and Kabiri Samani, A.R., “Critical submergence of vertical intakes using anti-vortex plates", Proc. of 6 Int. Conf. Civil Eng., IUT, Isfahan, Iran, 7, pp. 59-66, 2003. [19]

Yildirim, N., “Critical submergence for a rectangular intake", Journal of Engineering Mechanics, 130(10), pp. 1195-1210, 2004. [20]

Emdad, H. and Nikseresht, A.H., “Application of vortex lattice and quasi-vortex lattice method with free wake in calculation of aerodynamic characteristics of a hovering helicopter rotor blade in ground effect", Scientia Iranica, 10(1), pp.84-90, 2003. [21]

Kazemzade Hannani, S. and Parsinejad, F., “Incompressible stokes flow calculation using a finite point method", Scientia Iranica, 10(1), pp. 44-55, 2003. [22]

Ibrahim R. Teaima, Alla A.A. Gharieb and Younes M. A, “Feasibility of Direct Pumping for Irrigation Improvement Projects”, International Journal of Mechanical engineering and technology (IJMET), Volume 4, Issue 2, pp. 494-511, 2013. [23]

Launder, B. E. & Spalding, D. B., ”The numerical computation of turbulent flows”, Computation Methods, Application Mechanical, Engineering, 3, pp.269–289, 1974. [24]

Tokyay, T.E. and Constantinescu, S.G., “Coherent Structures in Pump Intake Flows: A Large Eddy Simulation Study”, 1st International Association Hydraulic Research Congress, Seoul, Korea, 2005. [25]

Karassik, I.J., Messina, J.P., Cooper, P. & Heald, C.C., “Pump Handbook”, McGraw-Hill, New York 2001. [26]

Dicmas, J.L., “Vertical Turbine, Mixed flow and Propeller Pumps”, McGraw-Hill, New York, 1987. [27]

Nagahara, T., Sato, & T. and Okamura, T., “Measurement of the Flow Around the Submerged Vortex Cavitation in a Pump Intake by Means of PIV”, Fifth International Symposium on Cavitation, Osaka, 2003. [28]