

Optimum Design of Pump Intake using CFD for Improving Hydraulic Performance

Elzahry Farouk Elzahry and Ashraf Ghanem*

Faculty of Engineering at Shoubra, Benha University, Egypt

*Corresponding author: eng_top@hotmail.com

Received 1 December 2019, Revised 31 December 2019, Accepted 14 January 2020.

Copyright © 2020 The Authors.

Abstract: A new El-Tabiya pumping station in Alexandria Governorate is considered in this work. The station consists of six axial pumping units. The old station is replaced by a new one and the old sump is reused. Pump operation depends not only on the efficiency of the pumping units but also on the proper design of the intake sump. Intake sumps receive water flowing from the intake channel and must direct it smoothly to the pump suction opening. If suction sumps are improperly shaped or sized, air-entraining vortices or submerged vortices are developed. The flow conditions at an entry to a pump depending upon flow conditions in approach channel, the various site-specific geometrical and hydraulic constraints, the location of pump intake with respect to the walls, velocity changes and obstructions such as piers, screens, etc., and rotational tendencies inflow produced upstream of the pump bays. The time and cost involved in-sump model studies for the design of sump geometry can be reduced to a large extent through Computational Fluid Dynamics (CFD) studies as the flow parameters can be predicted at the pump inlet with the change in geometry without actual running of the pump with CFD. Hence the design of the sump can be optimized to keep the flow parameters below limiting values. This study attempts to model the flow characteristic in a pump sump, minimize the swirl angles, increase the flow at the pump inlet and keep the flow parameters below limiting values. The numerical study carried out in this paper aims at optimizing the overall fluid flow in a pump intake by the use of a commercially available CFD code. CFD study was carried out on initial sump geometry and initial CFD results were analyzed.

Keywords: CFD; Design; Hydraulic; Pump intake; Vortices.

1. INTRODUCTION

Vortices appear in the pump sumps and they have negative consequences on facilities. Probably the most common problem and complicated to be solved is when there are multiple pumps and the flow inlet is very strong restricted due to context conditions (such as closed curves, reduced sections, etc.). The basic purpose of a pump intake is to supply water with a uniform velocity at the entry of an impeller. The fluid flow in pump intakes is rather complex involving expansions and turns together with fluid-structure interactions [1]. It is essential that the pumps get smooth swirl-free flow at their inlets while operating in such pump intakes. A proper intake design provides a uniform swirl-free flow to the pumps. Intakes of such pumps and the geometrical layout of the channel surrounding the pump bells are usually designed in an empirical fashion, relying on laboratory model studies and experiences with previous installations [2].

The Hydraulic Institute Standards specify general guidelines for the design of pump intakes. The site constraints usually call for a deviation from the Standards. It then becomes essential to investigate the pump intake to ensure a smooth flow over the entire flow range of the pumps and in all the combinations of the pumps. American National Standards for Pump Intake Design [3] recommended the standard sump design modification methods which are being widely used for optimizing the sump design to reduce the swirl angles and improve the flow velocity at the intake of the pump. Chen and Guo [4] presented a numerical model for three-dimensional turbulent flow in the sump of the pump station. A reasonable boundary condition for the flow in the sump with multi-intakes has been proposed, each of which may have different flow rates. Shabayek [1] developed an original model by the installation of sidewalls and curtain walls for the lateral distribution of flow within circulating water pump bays and to reduce the vortex activity in the vicinity of the pump such that the hydraulic conditions in the sump would meet the specified performance criteria. The use of a numerical approach starts with [5] in order to study the effects of non-uniform inlet flow on vortex generation and the effects of additional devices to prevent vertical flow formation. They have used a finite volume method to solve the RANS equations with the k -epsilon model. Takata *et al.* [6] reported large-eddy simulations of pump intake flows at low Reynolds number (10^4).

CFD benchmarks have been performed by Matsui *et al.* [7] in order to compare different software results with experiments. The advancement in high-speed computing and the development of robust turbulence modeling techniques as well as the

evolution of accurate numerical algorithms in the last few years have led to the development of CFD codes. These have provided an alternative means to study the hydraulic flow characteristics of pump sumps. There have been several studies [8-9] in the application of CFD to model flows in pump sumps. These studies have been conducted for idealized single intake rectangular geometry. Practical pump sumps, however, have multiple-intakes and the disturbances in the fore-bay diffusion area may be carried to the pump thereby altering the flow patterns and inducement of swirls at the pump inlet. While the studies [8-9] have identified the regions of vortex formation and have compared them with experiments and described swirl qualitatively (with streamlines), Chen and Guo [4] have conducted simulations in the pump sump where the fore-bay diffusion area was also modeled. The results were benchmarked by comparing the velocity profiles at various locations with the experimental results. The flow was qualitatively analyzed by studying the streamlines and vector plots at various locations in the pump sump. However quantitative results in terms of swirl angles at the pump inlet were not presented.

Desmukh *et al.* [10] have conducted simulations on multi- intake hydraulic structures and qualitatively analyzed the flow structure. However rigorous benchmarking of the simulation with experiments was not carried out. To do this, the conservation of mass and Navier-Stokes equations is normally used. The complexity of hydraulic conditions present within pump intake structures is such that it demands the full power of modern CFD to solve the equations of motion and turbulence models that involve multiple surfaces. Additional difficulties are associated with modeling free surface and vortex phenomena, the physics of which are not fully understood. Despite the prevalence of problems with pump stations, the application of numerical modeling to their design is limited (Leong *et al.* [11]). Luciano *et al.* [12] verified the ability of commercial CFD code to predict the formation of vortices in a pump sump with FLOW 3D using LES (Large Eddy Simulation). The free surface was tracked by means of a split Lagrangian method for the advection of the VOF (volume of fluid). The fluid was considered monophasic and incompressible. The numerical results demonstrated the capability of the model to identify the observed vortices in the physical model. The calculated vortex highest strength values were consistent with the air-entrained vortices. The floor vortex reached the highest vorticity magnitude and suggested the possibility of a cavitation core at the operating condition of lower submergence. Shukla *et al.* [13] used the commercial code CFX to carry out a two-phase flow simulation to capture air entrainment. They used an inflow solution. Turbulence was modeled by the k -epsilon model.

The paper deals with numerical and experimental results. The CFD model predicted the flow in sufficient detail to identify the locations, size, and strength of the vortices. The air entrainments and its location are well captured. The results are in accordance with the experimental investigation.

2. PROBLEM IDENTIFICATION

A new El-Tabiya pumping station in Alexandria Governorate consists of six axial pumping units. It is used to serve irrigation of 45000 feddans and consists of 6 pump units. Each pump unit is of discharge $7 \text{ m}^3/\text{s}$, total head of 5.5 m and motor power of 650 kW. The pumping station suction level varied from -4.5 m to -5 m. The main task of this work is to evaluate the hydraulic performance of the axial flow pumping stations when operating at low suction levels and evaluate the availability of this pumping station using CFD ANSYS software Ver. 18.1.

3. PUMP SUMP GEOMETRY

A study of the hydraulic problems of the suction basin of the new El-Tabiya pump station is essential due to the sediment problems that occur in the suction of pump intake. The worst case occurs when all units are in operation. The prototype layout includes a leading channel with difficult tendencies, approach channel, fore-bay, pump sump, and intake. It was proposed to carry out CFD analysis of the sump to see its overall suitability and for optimal operating conditions, in view of flow quality. As the flow quality was not good for initial sump geometry, several trials have been done for improvements. The flow study is done under consideration that pumps working at the minimum water level as the operation at a minimum water level are considered to be worst. Meshing is a crucial part of the analysis in which the whole geometry is divided into manageable shapes or elements, whose study is far simpler than the original body. These small units are studied individually and the compilation of these results gives the changes introduced in the domain. Elements of any shape can be used, for example, square triangle or tetrahedral. Here, 7217468 tetrahedral elements are used. The meshes have a total of 2757227 nodes. Nodes are the points on a meshed surface/body where the different elements meet. Figure 1 shows the surface, the volume mesh of the sump intake and the geometry.

4. APPLYING BOUNDARY CONDITIONS

The solver requires some initial values for initializing the finite element analysis procedure. These initial values are an approximation of the required conditions. The inlet boundary condition is applied at the entry in terms of total mass flow that is entering into the sump, using a high-intensity turbulence model. This model is chosen to keep the frictional, turbulence errors in consideration. At the outlet face named (OUTLET), an outflow is assumed, averaging over the entire face. There is no slip in the wall and the surfaces of the sump are kept smooth to reduce friction losses as much as possible. The domain type is a fluid domain with water as the flowing fluid in the domain. The standard k -epsilon turbulence model is used for the analysis. This is one of the most prominent turbulence models. The k -epsilon model has been implemented in most general-purpose CFD codes and is considered the industry standard model. The advection scheme used a high resolution. In this scheme, the blend factor values vary throughout the domain based on the local solution field in order to enforce a boundedness criterion. This scheme is a higher-order scheme and gives good results especially in the case of recirculation of flows. Mostly the solution is converged to 10^{-6} RMS level. The convergence of mass and momentum is ensured in the solution.

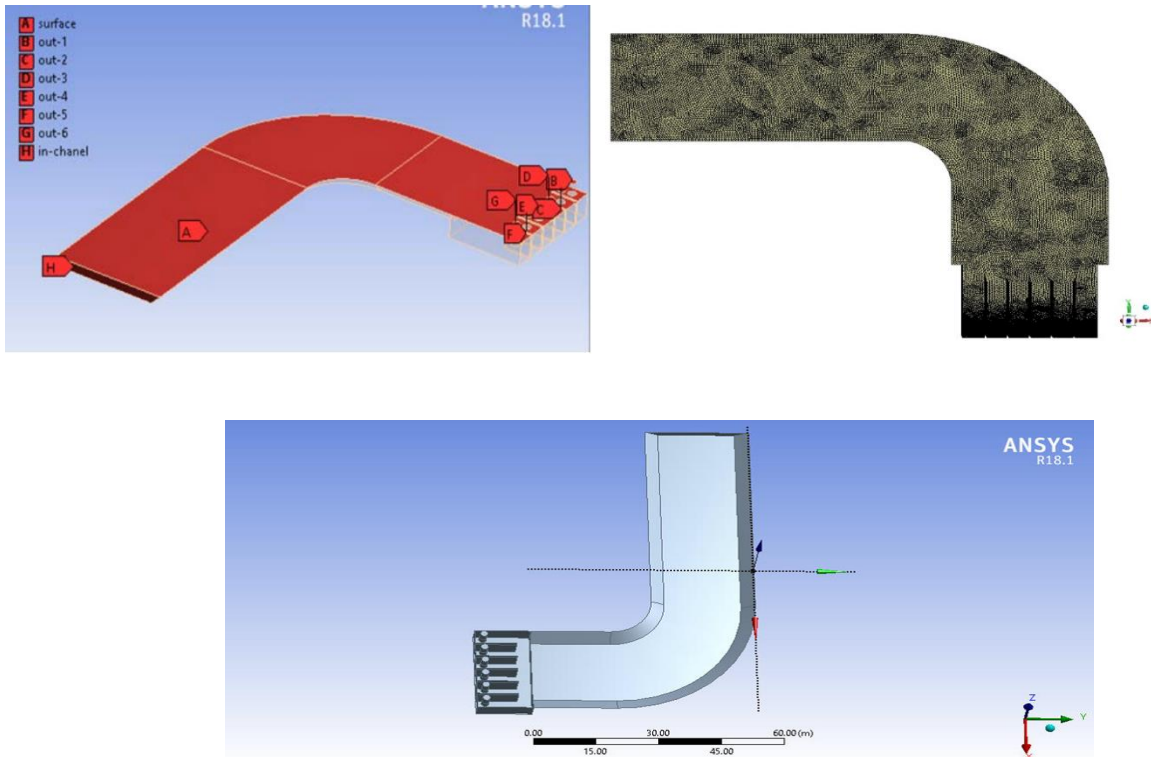


Figure 1. The Surface and the volume mesh of the sump intake

5. RESULTS AND DISCUSSIONS

Several operating cases were studied to find the best operating condition. They are divided into 4 cases.

5.1 Case 1: 6 Units with Suction Basin Height of 5 m

In this case, the height of the suction basin is decreased due to sedimentation that occurs throughout the year which causes the velocity to fall below the permissible limits and the flow velocity at the inlet suction of each pumping unit will become less than 0.3 m/s and this is not allowed. The shape of streamlines and its directions, as well as velocity contour, are identified for each case. When all six units are activated, it is found by displaying the streamlines to the lowest level of water in canal (plan1) that the streamlines shall be dense to the outer curve of the suction pipe as well as the direction of the drawing of the units (1, 2 and 3), The density of the streamlines decreases as the height of pump intake increases are shown in Figure 2(d). At the bottom level of the inlet pipes (plan2), the irregularity and distribution of the streamlines of the six units are also shown and their separation zones of the flow directions as shown in Figure 2(c). The velocity contour at the bottom level of the inlet pipes (plan2) shows the existence of a dead area with relatively low speeds as shown in Figure 2(e). It is also illustrated by the shape of the flow directions (vector) at the bottom level of the suction pipes (plan2). There is an effect of the vortices and separation zones of the flow directions, as shown in Figures 2(a) and 2(b).

5.2 Case 2: 6 Units with Suction Basin Height of 4 m

As can be seen from the results of the Case 2, there is a difference in the distribution of the streamlines between the units located on the outer and inner curve of the channel. The behavior of the streamlines is more uniform than its counterparts on the inner curvature. However, the regularity of the streamlines still has the insufficient, making the units not efficient as shown in Figures 3(c) and 3(d). The velocity contour at the bottom level of the inlet pipes (plan2) shows continuity of the existence of a dead area with relatively low speeds as shown in Figure 3(e). It is also illustrated by the shape of the flow directions (vector) at the bottom level of the suction pipes (plan2). There is an effect of the vortices and separation zones of the flow directions, as shown in Figures 3(a) and 3(b).

5.3 Case 3: 6 Units with Suction Basin Height of 3 m

The behavior of the streamlines is more uniform than its counterparts on the inner curvature in case one. However, the regularity of the streamlines started getting, making the units efficient compared to Case 1 as shown in Figures 4(c) and 4(d). The velocity contour at the bottom level of the inlet pipes (plan2) shows continuity of the existence of a dead area with relatively low speeds but less than case one as shown in Figure 4(e). It is also illustrated by the shape of the flow directions (vector) at the bottom level of the suction pipes (plan2). There is an effect of the vortices but it seemed less compared to case 1 and separation zones of the flow directions start to disappear around the suction pipe, as shown in Figures 4(a) and 4(b).

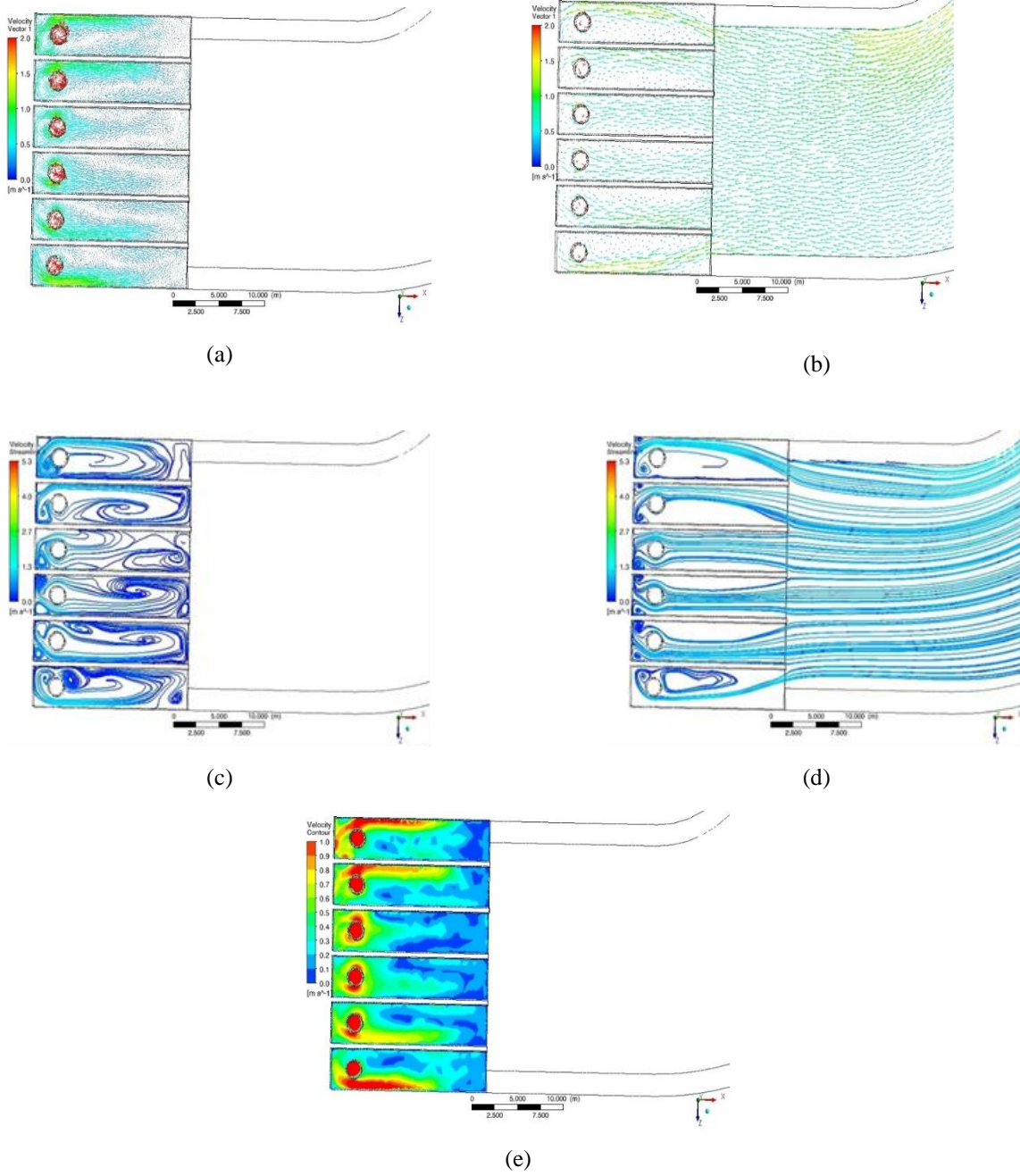
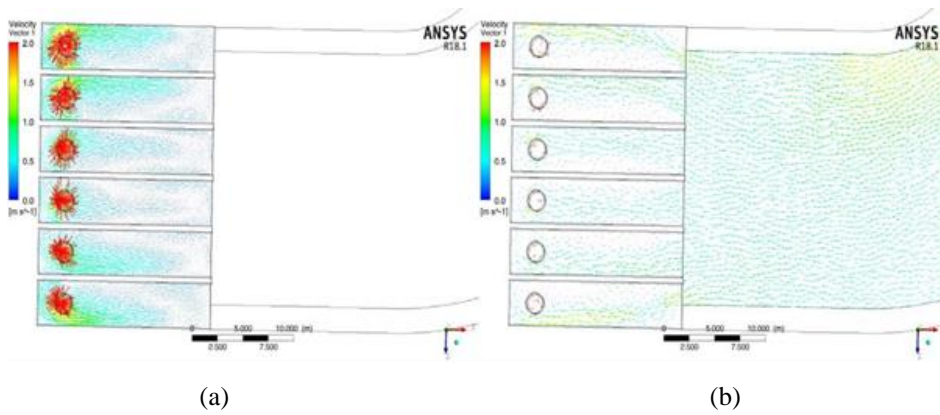
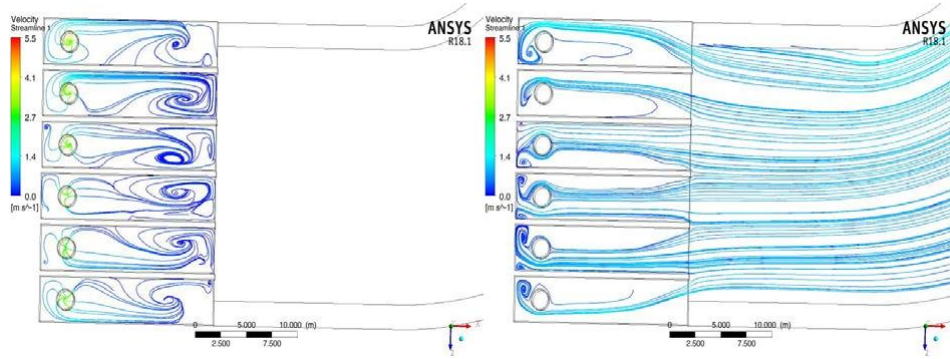


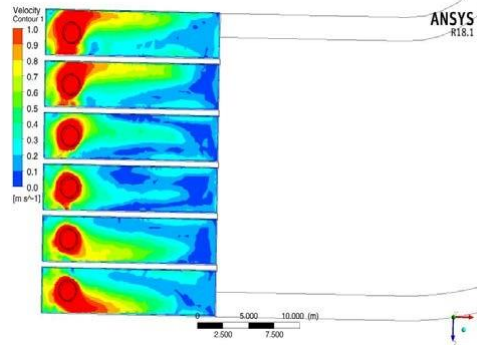
Figure 2. The vector distribution, streamline velocity and contour of velocity for Case 1





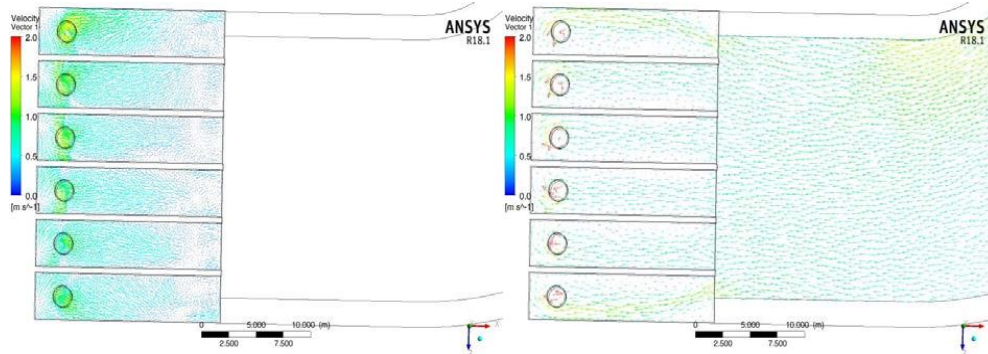
(c)

(d)



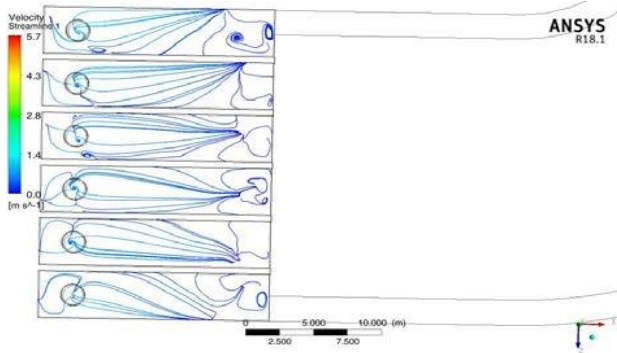
(e)

Figure 3. The vector distribution, streamline velocity and contour of velocity for Case 2

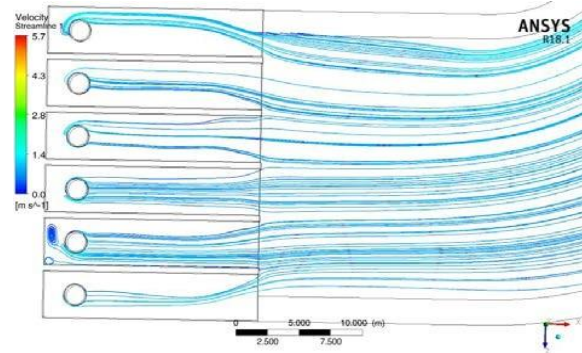


(a)

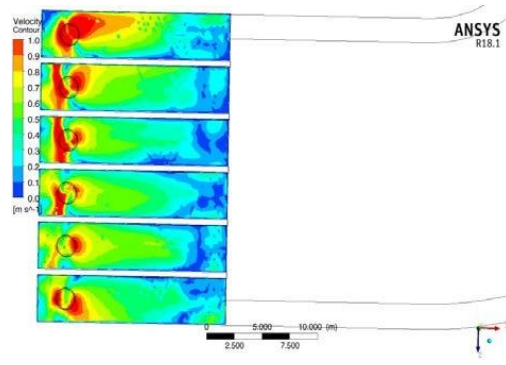
(b)



(c)

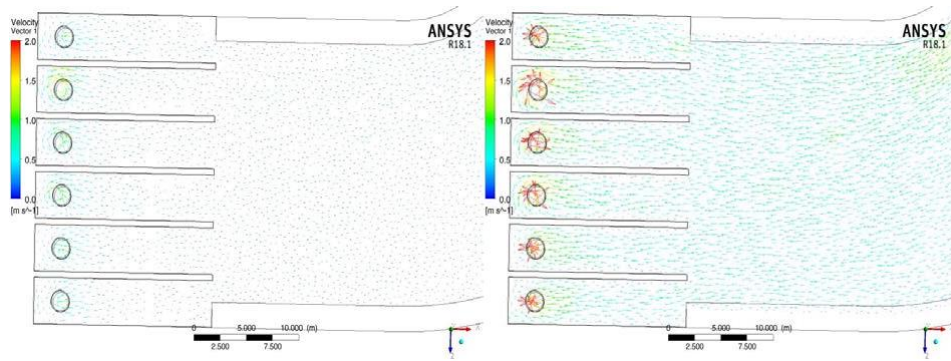


(d)



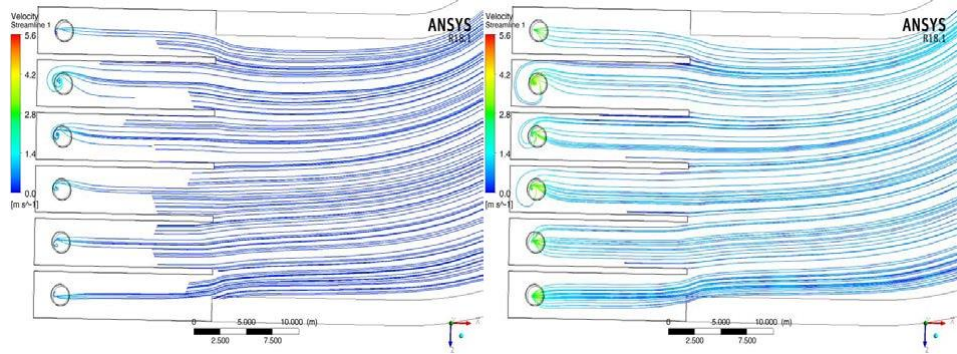
(e)

Figure 4. The vector distribution, streamline velocity and contour of velocity for Case 3



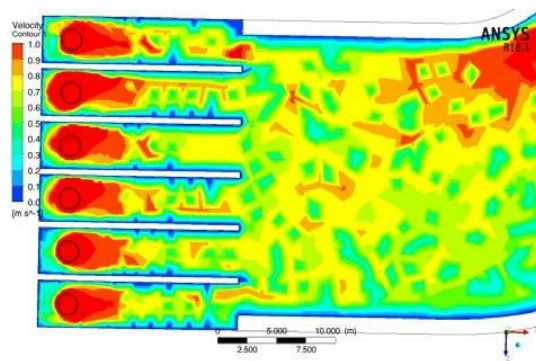
(a)

(b)



(c)

(d)



(e)

Figure 5. The vector distribution, streamline velocity and contour of velocity for Case 4

5.4 Case 4: The Canal Level Equal to the Pump Intake Level

This case is considered one of the best cases as a result of regularity streamlines and dense at a higher level of water. For the streamlines at the bottom level of the suction pipes, they show a good improvement in all units as shown in Figures 5(c) and 5(d). The velocity contour at the bottom level of the inlet pipes (plan2), the dead zone disappears with relatively high speeds as shown in Figure 5(e). It is also illustrated by the shape of the flow directions (vector) at the bottom level of the suction pipes (plan2). There is no effect of the vortices and no separation zones of the flow directions around the suction pipe regularity distributed, as shown in Figures 5(a) and 5(b).

6. CONCLUSION

From the previous study, it is clear that the problem of sedimentation that occurs in the suction of the new El-Tabiya pumping station leads to problems in the units and lack of efficiency and the occurrence of low speeds and the presence of dead zone and separation areas. To solve this problem, the level of pump intake must be raised to the same level of the canal.

REFERENCES

- [1] S. A. Shabayek, Improving approach flow hydraulics at pump intakes, *International Journal of Civil and Engineering, IJCEE-IJENS*, 10(6), 2010.
- [2] S. M. Borghei and A.R. Kabiri-Samani, Effect of anti-vortex plates on critical submergence at a vertical intake, *Transaction A: Civil Engineering*, 17(2), 89, 2010.
- [3] Hydraulic Institute, *American National Standard for Pump Intake Design*, Parsippany, NJ: Hydraulic Institute, 1998.
- [4] H. -X. Chen and J. -H. Guo, Numerical simulation of 3-D turbulent flow in the multi-intakes sump of the pump station, *Journal of Hydrodynamics: Series B*, 19, 42-47, 2007.
- [5] M. Tagomori and M. Gotoch, Flow patterns and vortices in pump-sumps, *Proceedings of the International Symposium on Large Hydraulic Machinery*, Beijing, 1989, pp. 13-22.
- [6] I. Takata, Y. Kawata, T. Kobayashi and Y. Morinishi, Large-eddy simulation of unsteady turbulent swirl flow in a pump intake, *Proceedings of the 1st European Computational Fluid Dynamic Conference*, Brussels, 1992, pp. 255-261.
- [7] J. Matsui, K. Kamemoto and T. Okamura, CFD benchmark and a model experiment on the flow in pump sump, *Proceedings of the 23rd IAHR Symposium on Hydraulic Machinery and Systems*, Yokohama, 2006, pp. 1(10)-(16).
- [8] M. Tagomori and M. Gotoch, Flow patterns and vortices in pump-sumps, *Proceedings of the International Symposium on Large Hydraulic Machinery*, Beijing, 1989, pp. 28-31.
- [9] W. L. Chuang and S. C. Hsiao, Three-dimensional numerical simulation of intake model with crossflow, *Journal of Hydrodynamics, Series B*, 23(3), 314-324, 2011.
- [10] T. S. Desmukh and V. K. Gahlot, Numerical study of flow behavior in a multiple intake pump sump, *International Journal of Advanced Engineering Technology*, 2, 118-128, 2011.
- [11] M. C. M. Leong and P. B. Clark, CFD modeling of pump sumps, *Proceedings of the 4th International Conference on Water and Wastewater Pumping Stations*, Cranfield, 2008, pp. 71-80.
- [12] C. Lucino and S. L. Gonzalo Duró, Vortex detection in pump sumps by means of CFD, *XXIV Latin American Congress on Hydraulics*, Punta del Este, 2010.
- [13] S. N. Shukla and J. T. Kshirsagar, Numerical prediction of air entrainment in pump intakes, *Proceedings of the 24th International Pump Users Symposium*, Texas, 2008, pp. 29-33.