ENHANCING HYDRAULIC PERFORMANCE AND ENERGY SAVING OF PUMPS USING COMPUTATIONAL FLUID DYNAMICS (CFD)

Sami A. A. El-Shaikh

ABSTRACT

The adverse flow conditions at pump intakes cause swirl and vortices reducing pump performance and efficiency. The flow at pump intake is affected by sump design and its associated flow conditions. This paper presents a numerical simulation using CFD technique at different conditions for predicting occurrence of vortices and swirl in a pumping station intake avoiding hydraulic problems, saving power consumption and maximizing efficiency.

Different approaches were simulated to find the optimum and reliable modification method to increase capacity, enhance hydraulic performance and decrease power consumption of the pumping station due to intake hydraulic problems. The results of the pumping station mathematical model in a good agreement with the physical model, A 20% increase in pump flow rate was obtained with no vortex and turbulence at the entrance of the pump by adding a cone under the bell mouth. The technique can be applied for future development of existing pumping stations with increasing water requirement without hydraulic problem in the pump intake, little energy consumption rate and high pump efficiency.

Keywords: Pump Intake, Vortex, energy, Computational Fluid Dynamics (CFD), Pump Hydraulic problems

1. INTRODUCTION & LITERATURE REVIEW

The efficiency and performance of pumping stations involving multiple pumping units depends not only on the efficiency of the pumping units but also on the proper design of the sump intake. A solution of site specific problems has been conducted for design of new sumps as well as for improving the design of existing sumps [1]. Researchers are now venturing into the field of numerical modelling of pump intake flows using (CFD). Flow conditions at entry to a pump depend upon flow conditions in approach channel, sump geometry, location of pump intake with respect to the walls, velocity changes and obstructions such as piers, screens etc. [2]. The pump intake was modified with a goal to eliminate air entrainment. An experimental model was built with these modifications and the air entrainment was seen to be eliminated [3].

A swirling flow occurs when pump-approach flow distributions within the intake bay are not uniform regardless of its origin. These problems encountered in the pump intake will affect the pump performance and significantly increase the operational and maintenance costs[4]. Different types of vortices are clearly observed. Flow pattern under several amount of discharge and submergence depth of suction pipe was monitored. In the range of tested discharge regarding to the kind of pump and electromotor, the flow pattern could not affect the energy consumption significantly [5].

Flow in a pump sump was modeled to evaluate the potential for the formation of vortices[6]. Several modifications to the intake design were done The objectives of the modifications were to improve 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 to meet the specified performance criteria [6]. The most common solution for avoiding air-entrainment is the use of anti-vortex devices and, especially, plates for large pipe or shaft intakes. If plates are used, then, the geometry and position of them should be studied experimentally [7]. Verifying the ability of a commercial computational fluid dynamic (CFD) code is done to predict the formation of vortices in a pump sump. It was intended to identify vortices of diverse origin and intensity in a geometrically simple pump sump of which experimental results under the same operating conditions are known [8]. A large amount of energy is consuming in pumping station for water supply. Inappropriate situation of pumping, and consequently more consumption of energy are related to intensity of disturbance of flow in pumping sump [9].

[Assistant Professor],[Mechanical & Electrical Research Institute, National Water Research Center, Ministry of Water Resources & Irrigation], [Delta Barrage, Egypt],[samiabd@yahoo.com]

The proper design of pump intake is not an easy task because of the various site-specific geometrical and hydraulic constraints. The time and cost involved in sump model studies for design and optimization of sump geometry can be reduced to a large extent through CFD studies.

2. MATHEMATICAL MODEL OF THE PUMP INTAKE

The governing four equations are as follows:-

$$V_{inlet} = \frac{Q_{inlet}}{A_{inlet}} \quad (1), \quad \theta = \tan^{-1}\frac{V_R}{V_A} \quad (2), \quad V_R = \frac{\pi d_v R}{60} \quad (3), \quad V_A = \frac{Q}{A} \quad (4)$$

Where,

V inlet = velocity of water at the inlet (m/s), Q inlet = flow rate / discharge (m3/s), A inlet= area of the inlet cross section depending on water level (m), V_R= Rotational velocity (m/s), VA= Axial velocity (m/s), dv=Diameter of vortimeter vane (m), R= Number of revolutions of the vane per minute (Rev/min), Q= Flow through the pump intake (m^3/s), A = Cross-sectional area of the intake (m^2)

Net mass flow out of control volume through control surface = Time rate of mass inside control volume

$$\underline{\textit{Transient}\ \text{term} + \text{Convection term}} = \frac{d\rho}{dt} + \nabla.(\rho v) = 0 \quad (5)$$

$$\frac{du}{dx} + \frac{dv}{dy} + \frac{dw}{dz} = 0 \quad (6)$$

$$\frac{d(\rho u)}{dt} + \nabla.(\rho u V) = -\frac{dp}{dx} + \frac{d\tau_{xx}}{dx} + \frac{d_{yy}}{dy} + \frac{d\tau_{xz}}{dz} + \rho f_x \quad (7)$$

$$\frac{a(\rho v)}{dt} + \nabla.(\rho u V) = \frac{d\rho}{dx} + \frac{d\tau_{yx}}{dx} + \frac{d\tau_{yy}}{dy} + \frac{d\tau_{yz}}{dz} + \rho f_y \quad (8)$$

$$K = \frac{3}{2}(U \times T_I) \quad (9), \qquad \in = C_u^{\frac{3}{4}} \frac{K^{\frac{3}{2}}}{b} \quad (10)$$

Where,

K = Turbulence kinetic energy, TI = Turbulence intensity, U = Inlet velocity, Cμ = Constant = 0.09, k = Turbulence kinetic energy, b = Characteristic length

3. CASE STUDY OF THE PUMP INTAKE&PROBLEM STATMENT

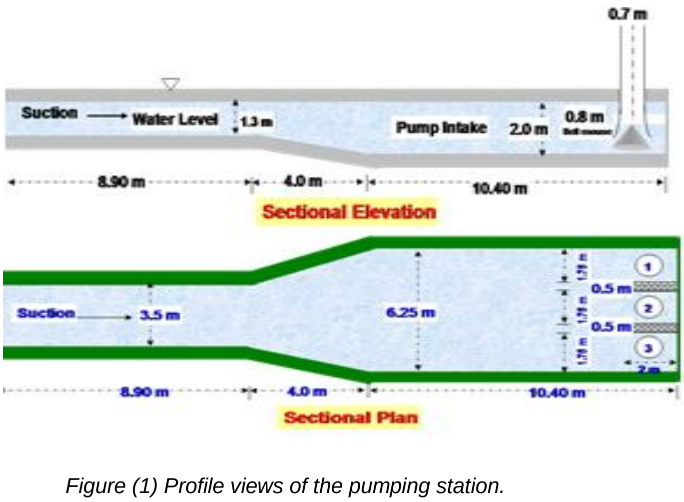


Figure (1) Profile views of the pumping station.

The pumping station consists of three pump units each pump with discharge 0.36 m^3/sec, head 6.2 m, rpm 970, and electric power 37 kW. Plan and profile views of the pumping station are shows in **Figure (1).** The measured discharge of the pump unit is 0.22 m^3/sec. There is decrease in the rated discharge about 30%, where eddies and turbulent were appeared in the intake. The energy consumption increase 10% this situation lead to drop efficiency 37% less than the rated efficiency.

4. NUMERICAL SIMULATION AND BUILDING A CFD MODEL

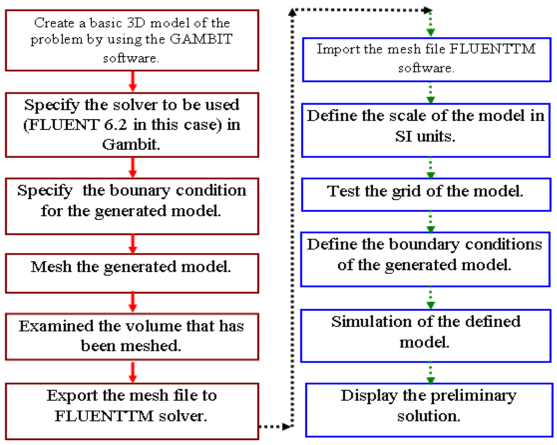


Figure 2. Problem solving steps in the CFD analysis

The simulation has been done using CFD code i.e. FLUENT software under ANSYS 13.0. The FLUENT model serves as a tool that discretizes and solves governing equations for specifics geometrics using a set of finite volume method. The problem solving steps in CFD analysis are summarized in **Figure 2.**

4.1 Geometry Design

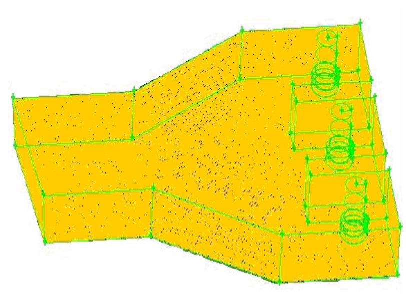


Figure 3. The surface and the volume mesh of the sump intake

The basic three-dimensional geometry is prepared using Gambit drawing. Total number of elements in the Sump is kept approximately 590 thousands. Where, the boundary conditions applied once similar condition as in the filed setup. The mesh model was divided into three sections. **Figure 3** shows the surface and the volume mesh of the sump intake.

4.2 Boundary Conditions&Initial Parameters

In CFD analysis, solution depends upon the appropriate boundary conditions. In case of sump, the outer wall is considered as a surface through which no flow can pass and the velocity at the surface is zero. These walls are defined as walls with no slip. The inlet boundary condition is applied at the entry in terms of total mass flow that is entering the Sump. The boundary conditions at the outlets were specified in terms of pressure boundary. The domain type is fluid domain with water as the flowing fluid in the domain. The standard shear stress transport (SST) turbulence model is used for analysis. Turbulent wall function is used as scalable.

5. RESULTS & DISCUSSIONS

Different approaches were simulated using CFD to find the optimum and reliable modification method to increase capacity of the pumping station and enhance the hydraulic and dynamic performance.

5.1. Case One: Existing Running Condition

In this case, flow measurement is 0.22 m^3/sec and inlet velocity is 0.35 m/sec. Three pump chambers with vertical pumps having an optimum suction bell diameter of 608 mm size were studied for nominated 0.36 m^3/sec flow. The chamber configuration available at existing site was used for this study. Minimum water level at 1.15 m (submergence level) above the sump floor was used in this study. The model results show strong vortices were created at the back, lift and right side of the pump as shown in **Figure (4)** entering the suction bell directly. The measured discharge of the pump unit is 0.22 m^3/sec. There is decrease in the rated discharge about 30%, where eddies and turbulent were appeared in the intake. The energy consumption increase 10% this situation lead to drop efficiency 37% less than the rated efficiency. The velocity distribution at the impeller location was also non uniform. These are the conditions which are harmful for the hydraulic performance as well as the mechanical performance of the pump. The standard [10] indicates that the inlet velocity mustn't increase than 0.3 m/sec at the input of the intake of the suction pipe of the pump even no swirling and vortices.

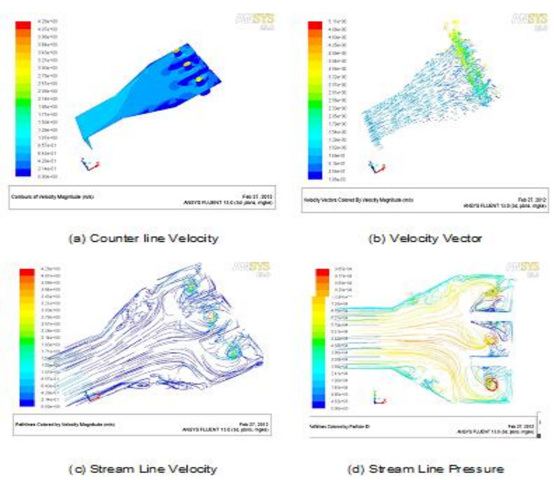
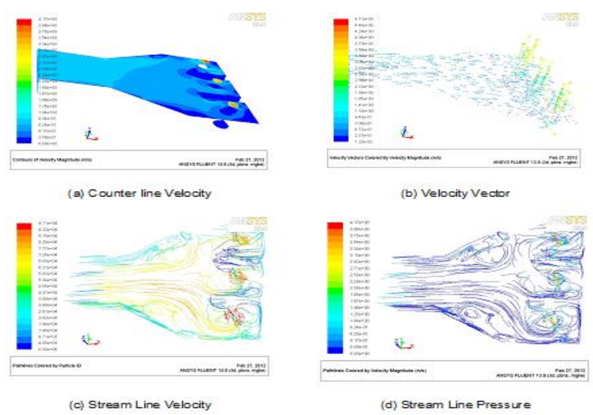


Figure (4) Case One: Velocity and pressure distribution under current running condition

5.2. Case Two: Added Cone under Bell Mouth



The same configuration was then used for flow with adding cone under bell mouth, where the measurement discharge is 0.22 m^3/sec before modification. The stream line plots for this case shows a uniform velocity distribution at the impeller location where the inlet velocity measured is 0.28 m/sec and flow rate is 0.36 m^3/sec which is the nominated discharge. Vortices of the second type, as defined by Hydraulic Institute Standard (HRI) were formed at the back side of the intake leading to a uniform flow velocity and stability of the hydraulic performance.

Figure (5) Case Two: Velocity and pressure distribution with Added Cone under bell mouse

Hence the suitability of existing chamber configuration was confirmed to flow of 0.36 m^3/sec. From this result, it can be concluded that a maximum flow of 0.36 m3/s can be expected from each pump without any hydraulic problems even at the minimum water level of 1.15 m above the sump floor. The model results that entering the suction bell directly isshown in **Figure (5).**

5.3 Case Three: Added Cone and Increase Discharge to 10%

The same configuration for second case was used but with increasing flow rate by 10% of the nominated discharge to become 0.4 m3/sec. The stream line plots still showed l a uniform velocity distribution at the impeller location where the inlet velocity measured is 0.28 m/sec. The model results that entering the suction bell directly shown in Figure (6). There is increase in the rated discharge about 10%, where eddies and turbulent were appeared in the intake. The energy

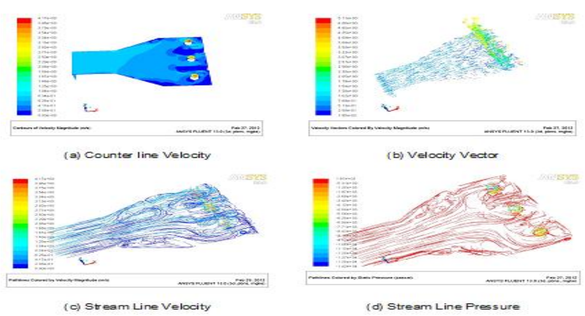
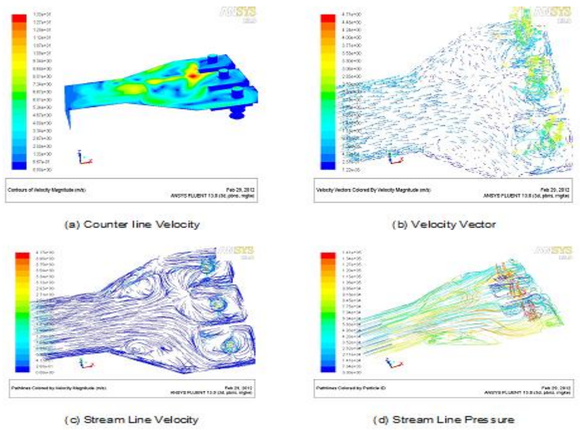


Figure (6) Case Three: Velocity and pressure distribution with Added Cone and increase discharge to 10%.

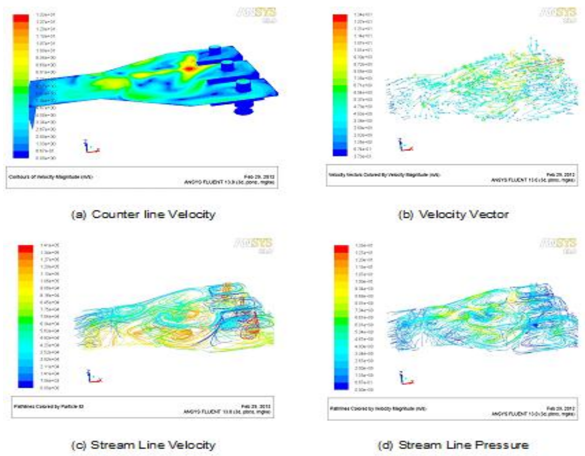
5.4 Case Four: Added Cone and Increase Discharge to 20%



The same configuration for second case was used but with increasing flow rate by 20% of the nominated discharge to become 0.43 m3/sec and the energy consumption slightly increase 2% as shown in **Figure (7).** The stream line plots for this still showed a uniform velocity distribution at the impeller location and higher than the third case where the inlet velocity measured is 0.295 m/sec. Velocity vectors and the recorded pressures in sump bottom show the effects of pumping discharge and submergence of suction pipe on flow pattern. In lower discharge and more suction pipe submergence, the differences from hydrostatic pressure are smaller.

Figure (7) Case Four: Velocity and pressure distribution with Added Cone and increase discharge to 20%.

5.5 Case Five: Added Cone and Increase Discharge to 30%



The same configuration for second case was used but with increasing flow rate by 30% of the nominated discharge to become 0.47 m3/sec. The stream line plots for this showed no uniform velocity distribution at the impeller location but higher than the all cases before where the inlet velocity measured is 0.37 m/sec (higher than first case) and vortices and swirling returned to appear at the entrance of the pump. The energy consumption increase 8% this situation lead to drop efficiency 15% less than the rated efficiency.

Figure (8) Case Five: Velocity and pressure distribution with Added Cone and increase discharge to 30%.

So, from the five cases itwas found that the forth case is the best solution where it gives the best results by adding cone at the entrance to the pump with increasing flow rate by 20%.

CONCLUSIONS

The research gives different scenarios to modify and control hydraulic problems in the pump intakes.

The research proves that adding a cone down under the bell mouth improves the dynamic and hydraulic performance of the pump, the flow rate increase 20% without swirling and turbulence. Power consumption rate is decreased of the pump efficiency is decreased.

This technique can be relied on in the simulation real sumps (multi of scenarios for the shape and movement of water at the entrance to the pump) rather than physical modeling in order to reduce costs and safe time and efforts.

The results help the decision makers of the pumping station for future water requirement and development with assuring safe and reliable running condition of the pumping station.

REFERENCES

1. Sehloff A. P., Curtis G., 2006. Pumping Station Modifications to Comply with Ansi/Hi 9.8 Improve Performance While Lessening O&M Requirements - two Case Studies. Water Environment Foundation, All Rights Reserved.
2. DesmukhT. S., GahlotV.K, July 2010. Simulation of Flow through A Pump Sump and its Validation. IJRRAS 4 (1).
3. shuklaS. N.,Kshirsagar J. T., 2008. Numerical prediction of air entrainment in pump intakes. Proceedings of the Twenty-Fourth International Pump Users Symposium.
4. RozainyM. R.,Abustan I., Abdullah. M. Z., M. Ashraf M. I., 2008. Application of Computational Fluid Dynamics (CFD) in Physical Model of Pump Sump to Predict the Flow Characteristics. ICCBT 2008 - D - (07) – pp79-90, ICCBT.
5. Müller M., 2009. Vortices at intake works of pump-storage schemes 6th International Symposium on Ultrasonic Doppler Methods for Fluid Mechanics and Fluid Engineering.
6. Shabayek1 S. A., Nov. 2010. Improving Approach Flow Hydraulics at Pump Intakes. International Journal of Civil & Environmental Engineering IJCEE-IJENS Vol: 10 No: 06, IJCEE,.
7. BorgheiS.M., Kabiri-SamaniA.R., April 2010. Effect of Anti-Vortex Plates on Critical Submergence at a Vertical Intake. Transaction A: Civil Engineering, Vol. 17, No. 2, pp. 89{95, c Sharif University of Technology,.
8. LucinoC., LisciaS. y. DuróG., November 2010. Vortex Detection in Pump Sumps by Means of CFD. Xxiv Latin American Congress on Hydraulics Punta Del Este, Uruguay, (Iahr),.
9. Rizi1A. P., 2009. Experimental Investigation of Relationship between Sump Flow Pattern and Pumping Energy Consumption. International Symposium on Water Management and Hydraulic Engineering, Ohrid/Macedonia, 1-5 September.
10. Hydraulic Institute, 1994. Hydraulic Institute standards for centrifugal, rotary, and reciprocating pumps. 18th edition, Cleveland, Ohio.