Predicting and Improving Hydraulic Performance of Pumping Suction Intakes By Computational Fluid Dynamics (CFD).

***Ashraf Ghanem, Gamal Helmy Elsayed, Mohamed Fayek Abd Rabbo, Mostafa Abuzeid, and Elzahry Farouk Elzahry.**
Faculty of Engineering at Shoubra, Benha University, Egypt.
*(eng_top@hotmail.com)

Abstract

The disturbance in velocity distribution generates hydraulic instability of pumping units, which leads to failure, damage and other operation and maintenance difficulties. The objective of this research is to predict and enhance the hydraulic problems in the sump intake of FARASKOUR drainage pumping station on faraskour drain Damietta governorate. The first and fifth units of the operating, water were not analyzed, study of the hydraulic problems of the suction basin of the new pump station due to the sharp rotation in the suction guide from the sharp rotation to the quay station and the continuous discontinuation of the first and fifth units due to the lack of regular water entering the unit, and recommend the suitable modifications to eliminate the operations and maintenance costs.

A numerical simulation was done to investigate the hydraulic stability of the station. There are four cases were done. The first case, when all units "five units" operates at the same time, the second case when three units operate "1, 2 and 3", the third case when other three units "3, 4 and 5" operates, and the last case when three units operate "1, 3 and 5". The ANSYS R18.1 flow simulation software, Computational Fluid Dynamics (CFD) is used to simulate the flow conditions at different working pumping units and different water levels to predict the hydraulic problem at the suction side.

The results indicated that the problem of flow is due to the sharp bending of the suction channel at the entrance of the units to the station, leading to two negative phenomena, the first one not to distribute well to the incoming lines of the units, and the second is appearance of places of relatively low speeds. The internal bending of the stream just before the entry of units directly to this area negatively affects the work of units (4, 5) and the problem becomes more specific with the formation of depositions.

In general, the results indicated that with five pumping units' scenarios in operation appear dead zones in the inner curvature of the intake. From simulation results, the geometry of the intake is proper for running four parallel flow pumps with the designed flow rate and reduces the distance from both sides of the canal (make Protection basin).

Keywords: Hydraulic stability; Vibration level; CFD; Velocity distribution; sump geometry.

1. Introduction:

In Egypt, there are more than 1,500 irrigation and drainage pumping station. These pumping stations exposed many problems such as hydraulic, dynamic and electrical, which affect the performance, safety, and efficiency of the pumping stations. These problems are costing the country millions of pounds of foreign currency for operation and establishment, maintenance, and purchase of spare parts for these stations to maintain the distribution of water in Egypt [**1**]. It is requiring a continuous assessment, surveillance, monitoring and analyzing the performance and avoids the problems that accelerate the collapse and damage the component of these pumping stations. It is an accepted fact that the faulty design of pump sump or intake is one of the major causes of the unsatisfactory operation of pumps in any pumping station. There are many investigators predicting the proper design of the intake pumping station. The flow conditions at the 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., and rotational tendencies inflow produced upstream of the pump bays, Desmukh, and Gahlot **[2]**. A number of deficiencies and problems with vertical pumps are often related to sump design rather than mechanical imperfections. When a pump intake bays are not properly designed, severe swirling flow problems may occur in the pump bay, Mohd Remy et al **[3]**. One of the sources of a disturbance at intakes is the existence of free-surface vortices with an air core. The most common solution for avoiding air- entrainment is the use of anti-vortex devices and, especially, plates for large pipe or shaft intakes, Borghri and Kabiri **[4]**. The flow characteristics of pump sump and performance analysis of the mixed flow pump should be done. The efficiency of anti-vortex devices to insure the uniform flow is confirmed. From the numerical analysis, the inception of cavitation is observed on the suction surface where the leading edges meet the tip, and then the cavitation zone expands, Y X Zaho et al **[5]**. Numerical analysis for the flow characteristics of a sump model with pump intake and good agreements was achieved by comparing the numerical results with the experiments, Rajendran et al **[6]**. Submerged vortex is introduced numerically by analyzing the flow in the pump sump with and without baffle plates, Iwano et al **[7]**. A multi-intake pump sump model analyzed by using CFD analysis to check the flow uniformity by predicting the location, number, and vorticity of the vortex, Lee et al **[8]**. A design guideline for the shape of pump sump is suggested by model test and CFD analysis, Lee **[9]**. Moreover, the Turbo-machinery Society of Japan **[10]** has revised the standard of pump sump model test and the revised standard examined the possibility of numerical analysis for the prediction of the flow in the sump model using several commercial and in-house CFD codes. The flow uniformity according to the flow distribution in the pump intake channel is examined to find out the cause of vortex occurrence in detail by experiment and CFD, Kim et al **[11]**. water levels variation in the sump intake is studied in El- SHABAB Pumping Station. ANSYS Ver.17.1 flow simulation software is used to simulate the flow conditions to overcome decreasing water level in the low demands period, winter closure. Seven scenarios are studied to obtain the best one to apply it to that condition **[13]**.

2. Problem Identification and methodology

New Faraskour drainage Pumping Station located at faraskour drain Damietta governorate. it consists of five axial pumping large units with the discharge of 5 m3/sec for each unit. The pumps are driven by a 650 kW electric motor with a gearbox. Figure (1) shows the New Faraskour drainage Pumping Station. The operating system depends on running not more than four pumping units.

Figure 1: New Faraskour drainage Pumping Station.

ANSYS 18.1 (CFD) software package was used to solve the flow equations using Navier Stocks Equations within the suction and pump inputs using the standard K and Epsilon equations. A three-dimensional model was designed for the station's discharge pipe, impeller, pumps, and five intakes for five vertical pumps.

Several scenarios for the operation of stopping units and operating other units to reach the best scenarios for the operation of units within the station, to identify and analyze the problem to select the best scenarios to control flow within the stream and entrances units. Figure (2) shows a two-dimensional image of the suction sump. The three-dimensional unstructured mesh (tetrahedral) has been created by using the structured mesh (hexagonal).

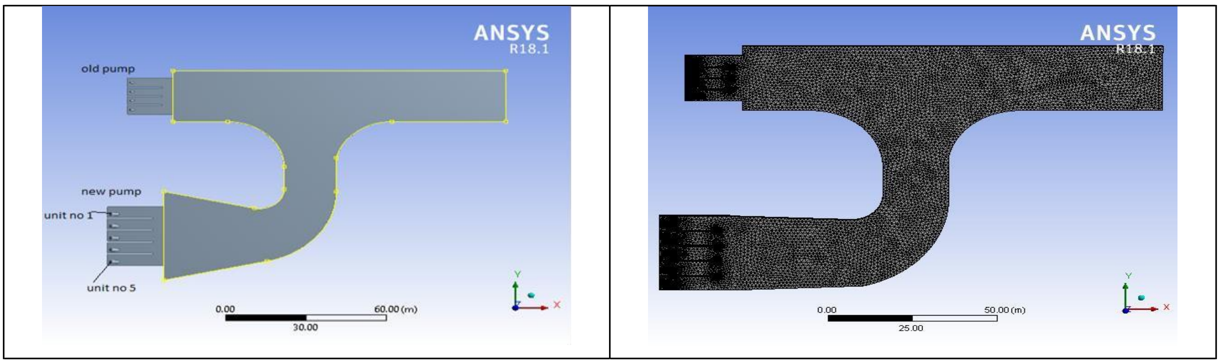


Figure 2: Two-dimensional image of the suction sump and unit input.

3. Boundary conditions and Grid Generation

To get an accurate simulation, the outlet volume flow applied in these two cases. The numerical analysis is a transient state. The "ideal wall" condition simulates the level of water, while the environment pressure condition is applied to simulate on the "inlet caps".

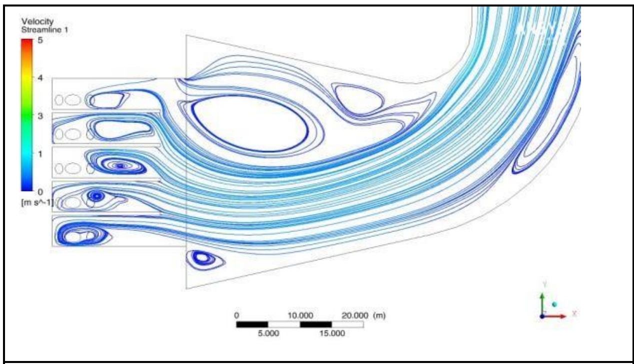
The quality of the computational mesh has an important role in achieving the desired accuracy of the simulations especially if the computational domain is very complex. The basic three-dimensional geometry is prepared using The ANSYS R18.1 software flow simulation software drawing with the grid surfaces plot of a sump. The equations are supplemented by fluid state equations defining the nature of the fluid, and by empirical dependencies of fluid density, viscosity and thermal conductivity on temperature. To predict turbulent flows, the Favre-averaged Navier-Stokes equations are used, where time-averaged effects of the flow turbulence on the flow parameters are considered, where the other, i.e. large-scale, time-dependent phenomena are taken into account directly. Three-dimensional unstructured meshes are used for the flow simulation in the pump sump. The unstructured mesh is used for this study due to model complexity and easy to mesh especially at the intake section. The numerical solver uses unstructured meshes that allow flexibility in meshing very complex geometries while maintaining high-quality computational mesh which is necessary for obtaining accurate solutions **[12]**.

4. Results & Discussions for Existing Operating Condition

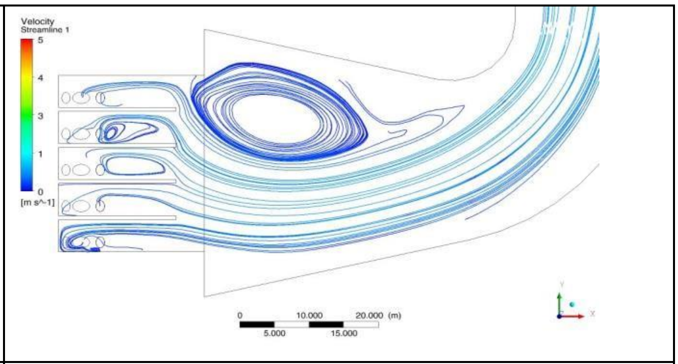
The existing operating condition contains four scenarios; these Scenarios were done to describe the shapes of the flow, its lines, and directions, as well as forms of speeds for every four scenarios to see the best-operating conditions for the units as follows. Then the same four scenarios were done but after reduces the distance from both sides of the canal (use Protection basin), close to the end of the suction channel to redistribute the lines flow within the stream at the entrance of units.

4.1. The first case (all five units operating)

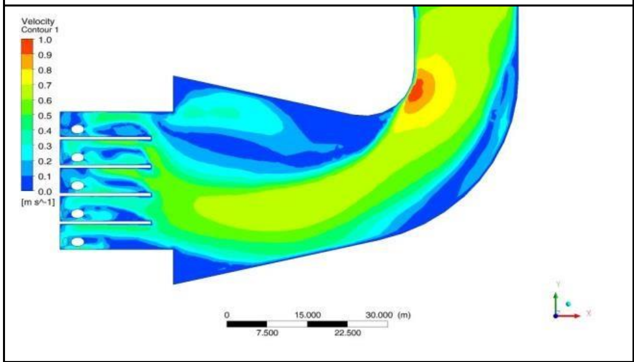
For the first scenario, when all five units are operated, it is found that through the width of the flow lines to the maximum level of water, the free surface, the flow lines are dense with the external bend of the suction path shaft (through units 3, 4 and 5), while it decreases with the internal bend of the suction path Especially unit 1, and appear of separation zone at the external bend as shown in figure (3-a) which shows the flow lines in the suction basin and the intake of the pumps. Also at the level of the bell, it is found that irregularity for distribution of the internal flow lines of the five units and are dense in the direction of the outer curvature of the pipe and less of the external bend, as shown in **figure (3-b)** which shows the distribution of the internal flow lines of the five units and showing their density in the direction of the external bend of the course. It is also a velocity contour of the channel and the entrance shows a relatively dead zone with relatively low speeds towards unit 1, as shown in **figure (3-c)**. In another hand, it’s found that from the shape of flow directions (vector) at the bottom level of the pipes of the pull a clear effect of the swirl (whirlpools) and separation zones of the flow directions (especially the unit number 1 and 5), as shown in **figure (3-d)**.



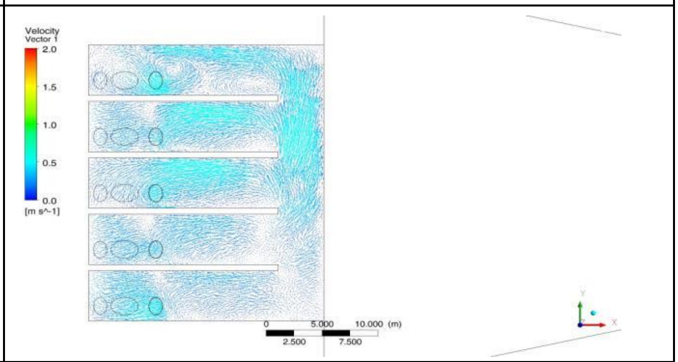
(a) Flow lines in the suction basin and the intake of the pumps.



(b) Distribution of the internal flow lines of the six units.



(c) Map of velocities of the channel and the entrance.



(d) The shape of flow directions at the bottom level of the pipes.

Figure 3: **First case: all five units operating**

4.2. The second case (three units operating (3&4&5))

From the results of the first scenario, it's clear that there is a difference in the distribution of the flow lines between the units located on the external and internal bending side of the sump. Therefore, we suggested doing another scenario by running the units separately to observe the flow distribution. In the second scenario when three units are operated (units 3,4,5 which located in the outer bend of the suction path), it is found that the behavior of the flow lines is more uniform in the outer bend of the suction path than that of the internal bend, swirl were appeared at the entrance of units (1, 2), as shown in figure (4-a) which shows the flow lines in the suction basin and the intake of the pumps. Also at the level of the bell, it is found that the direction of the flow lines is the regular distribution for units (3,4,5), and the swirls were appeared at the units (1, 2), as shown in figure (4-a) which shows the flow lines for different units in water level. However, a velocity contour shows that there are dead zones of the velocity values on the inner bend of the intake, as shown in figure (4-b). While from the flow directions at the level of the bell it appears that the presence of places of separation and swirls at unit 5.

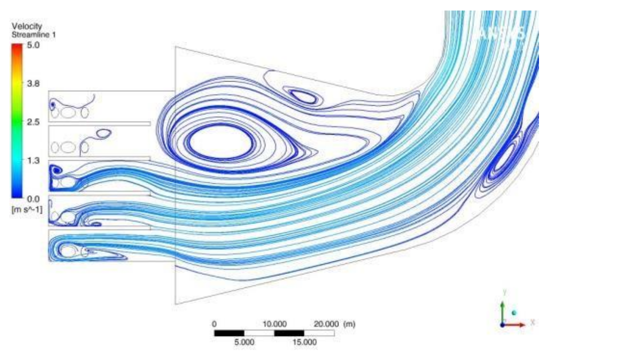
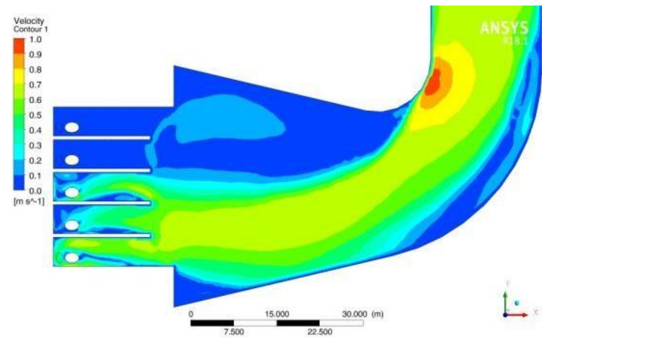
	
(a): Flowlines for different units at the level below the bell level.	**(b)** Map of velocities the highest level of water.

Figure 4: **Second case (three units operating (3&4&5))**

4.3. The third case (three units operating (1 &2&3))

In the third scenario when three units are operated (units 1,2,3 which located in the inner bend of the suction path), it is found that irregularity the lines of flow at the maximum level, opposite direction for lines of flow far from the units (1, 2, 3), effect of the sharp curvature of the stream in the form of flow lines where it’s disappearing at units (1, 2), as shown in figures (5-a). While, at the level of the bell it’s clear the effect of turbulent flow by the sudden change in the direction of the flow lines before the intake of the working units and vortices in unit 1, as shown in figure (5-a). However, a velocity contour shows irregularities of the speeds at the intake of the three units, as shown in figure (5-b). Finally, from the flow directions at the level of the bell, it appears the presence of separation zones and vortices at the bell level for the three units.

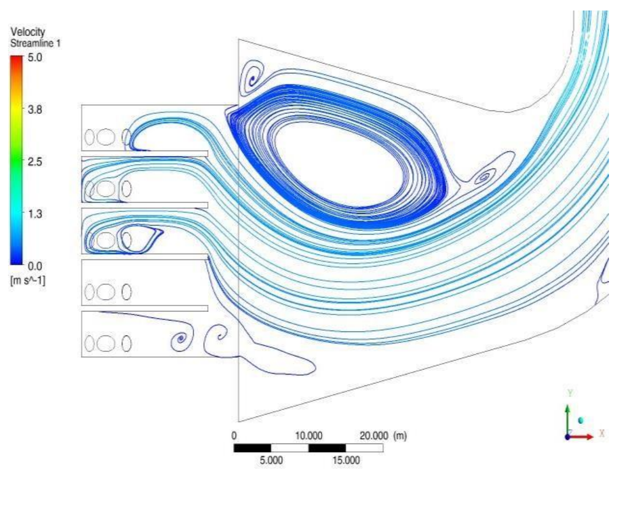
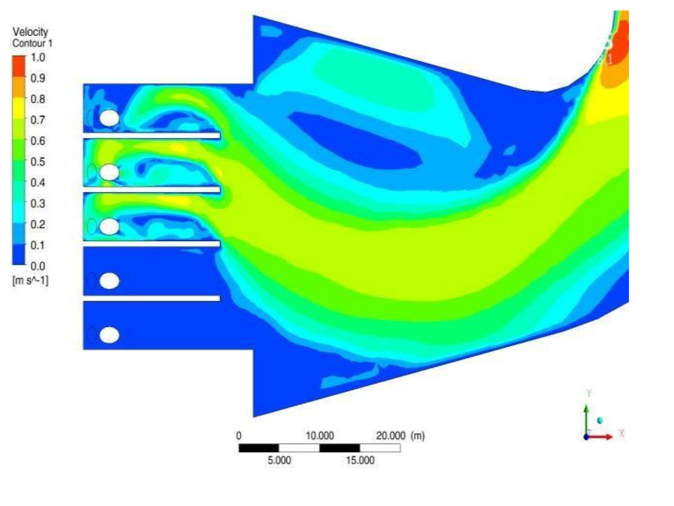
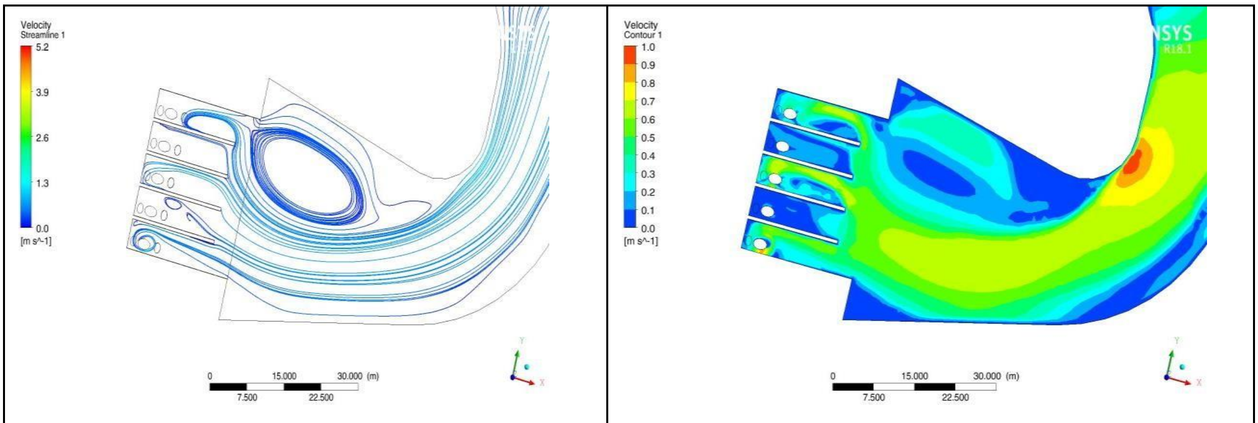
	
(a) Flow lines at the bell level.	**(b)** Map of velocities with the maximum water level.

Figure 5: **Third case (three units operating (1 &2&3))**

4.4. Forth case (four units operating (1&3&5))

In the fourth scenario when three units are operated (units 1,3and 5), it is found that the flow lines were not distributed regularly on the bending and at the entrance of the inputs unit, decreasing density of the flow lines in the unit, as shown in figure (6-a). However, a velocity contour shows indicate that still irregular velocities within the entrances of the working units, as shown in figure (6-b). While from the flow directions it appears that there are areas of separation of the flow and the effect of the whirls at the level of the bell, especially for unit 1.



(a) Flow lines at the bell level for the fourth scenario.	**(b)** Map of speed velocities for the fourth scenario.

Figure 6: **Forth case (four units operating (1&3&5))**

From the results above there are two problems were occurred, the first problem is the poor distribution for the flow lines of the water at the entrance to the units due to a sharp bend in the suction duct that impedes the stability of flow in the entrance of the units, especially when units (1, 5) run together at the same time. Where the second problem is the presence of areas where velocities are relatively low so, it becomes places of accumulation of sediments, which affects the work of the units. Therefore, a redistribution of the flow lines in the suction basin and in the entrance of the units was suggested to solve these problems. The first solutions reduced the distance from both sides of the canal (make Protection basin), to regulate and redistribute the flow lines at the entrance of units as shown in figure (7).

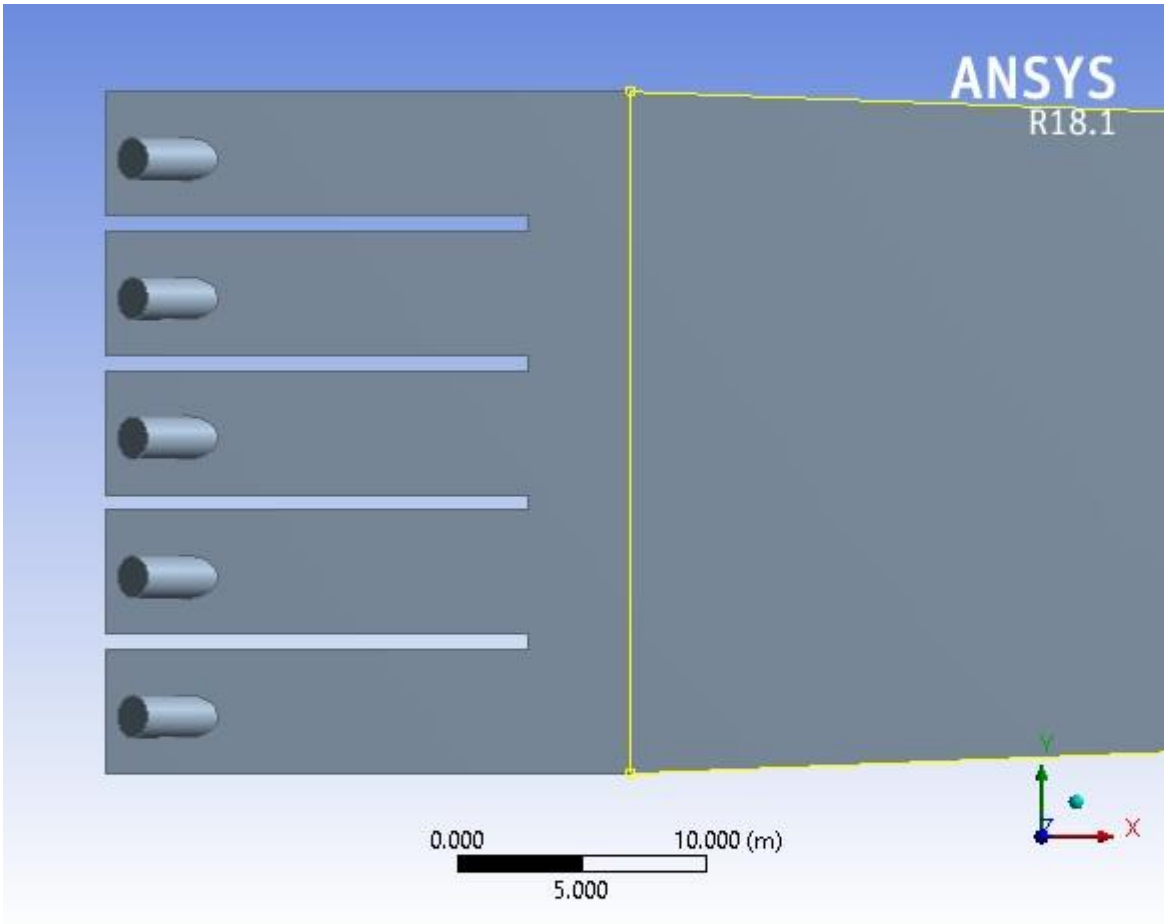
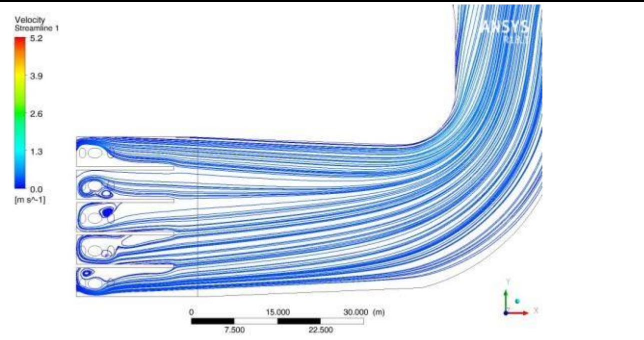


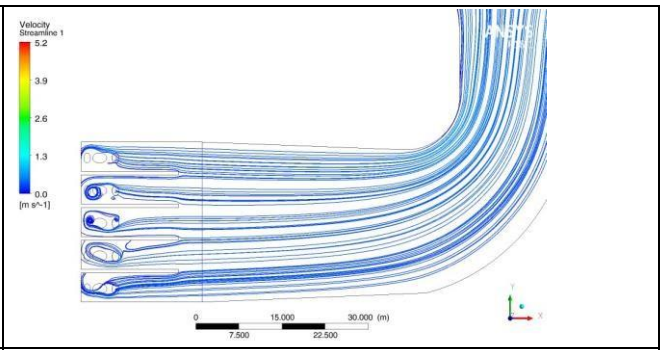
Figure 7: Setting the proposed routers through the suction sump

5. ***Results and discussion for suggested the problem by (*****make Protection basin).**

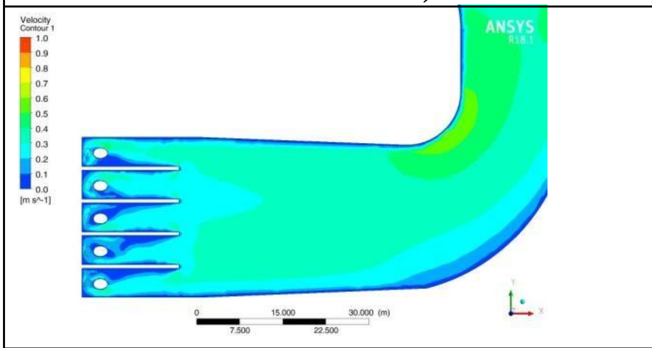
The results of the flow lines of the first scenario which is the worst which the operation of 6 units at the same time, after reduced the distance from both sides of the canal (use Protection basin) show good redistribution of the flow lines at the entrance of the units, especially at units (1 & 5) which had the flow lines, as shown in Figure (8-a). In addition, it is appearing that a good distribution of the internal flow lines for all five units working, especially the units that were previously defective, namely the units (1, 5), as shown in Figure (8-b). And also, a velocity contour shows velocity becomes uniform in the intake and at the entrance unit, as shown in Figure (8-c). Finally, the flow directions at the level of the bell show decreasing density separation places and vortex locations below the units, as shown in Figure (8-d).



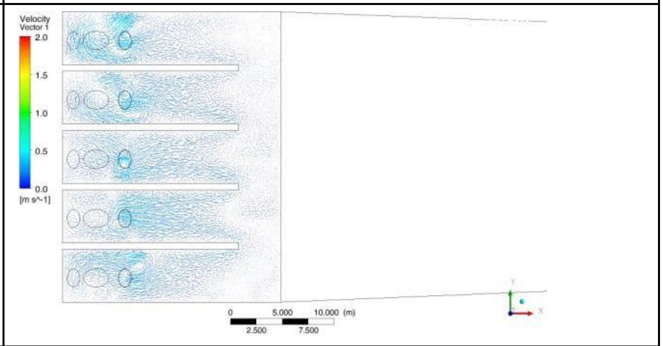
(a) Flow lines at a maximum level after (using Protection basin).



(b) Flow lines at the level of the bell after (using Protection basin).



(c) Map of velocities after (using Protection basin).



(d) Flow directions at the bell level after (using Protection basin).

Figure 8: **First case: all five units operating**

6. Conclusions

1. The technique of CFD plays an important role in predicting flow characteristics in the intakes of pumping stations of efficient hydraulic performance and in both rehabilitation and constructing new pumping stations.
2. This technique can be relied on in the simulation of real sumps rather than physical modeling in order to reduce costs and save time and effort.
3. The research indicates that the problems of flow in the new FARASKOUR drainage pumping station are due to the sharp bending in the intake at the entrance of the units to the station.
4. The problem of flow is due that bad distribution for the incoming flow lines of the units leads to the low intensity at the internal bending especially at unit no 1 and increase at the external bend at the unit no 5. This problem affects the work of the units especially units (1, 5) when operates together.
5. two scenarios with different four cases for every scenario were done to describe the shapes of the flow, its lines, and directions, as well as forms of speeds for every five scenarios to see the best-operating conditions for the units as follows, to determine the performance of the units under different circumstances.
6. The first scenarios with different four cases for every scenario were done to determine the performance of the units under different circumstances.

7. From simulation results, the geometry of the intake is proper for running four parallel axial flow pumps with the designed flow rate and after reduced the distance from both sides of the canal (use Protection basin).

7. References

[1] S. M. Abdel-Rahman and Sami A. A. El-Shaikh, "Diagnosis Vibration Problems of Pumping Stations: Case Studies", 13th International Water Technology Conference, Hurghada, Egypt, pp.419-434, 2009.

[2] Tanweer S. Desmukh and Gahlot "Simulation of Flow through a Pump Sump and its Validation", IJRRAS 4 (1), pp.7-17, July 2010.

[3] Mohd. Remy Rozainy and M. I. Mohd. Ashraf, "Application of Computational Fluid Dynamics (CFD) in Physical Model of Pump Sump to Predict the Flow Characteristics", ICCBT08, D - (07) – pp.79-90, 2008.

[4] S.M. Borghei and A.R. Kabiri-Samani, "Effect of Anti-Vortex Plates on Critical Submergence at a Vertical Intake", Transaction A: Civil Engineering, Vol. 17, No. 2, pp. 89-95, Sharif University of Technology, April 2010.

[5] Y X Zhao1, C G Kim1, and Y H Lee, "CFD Study on Flow Characteristics of Pump Sump and Performance Analysis of the Mixed Flow Pump", 6th International Journal of Pumps and Fans with Compressors and Wind Turbines,2013.

[6] V. P. Rajendran, S. G. Constantinescu, and V. C. Patel, "Experiments on Flow in Model Water Pump Intake Sump to Validate a Numerical Model", Proc. of ASME Fluids Engineering Division Summer Meeting FEDSM (Washington, USA, 21–25 June 1998), pp. 21-25, 1998.

[7] R. Iwano, T. Shibata, T. Nagahara and T. Okamura, "Numerical Prediction Method of a Submerged Vortex and Its Application to the Flow in Pump Sumps with and without a Baffle Plate", Proc. of the 9th Int. Symp. on Transport Phenomena and Dynamics of Rotating Machinery (Honolulu, Hawaii, USA, 10-14 February 2002) 1-6, 2002.

[8] J. W. Choi, Y. D. Choi, C. G. Kim, and Y. H. Lee, " Flow Uniformity in a Multi Intake Pump Sump Model", Journal of Mechanical Science and Technology 24 (7) 1389-1400, 2010.

[9] Y. H. Lee, "Establishment of Design Guideline for the Pump Intake Shape using the Result of Model Test", R&D Report, K-water (in Korean), 2004.

[10] Turbomachinery Society of Japan, "Standard Method for Model Testing the Performance of a Pump Sump", TSJ S002, 2005.

[11] C. G. Kim, J. W. Choi, Y.D. Choi, and Y. H. Lee, "A Study on the Effectiveness of an Anti-Vortex Device in the Sump Model by Experiment and CFD", 26th IAHR Symposium on Hydraulic Machinery and Systems, 2012.

[12] B. E. Launder and D. B. Spalding, "The Numerical Computation of Turbulent Flows", Computer Methods in Applied Mechanics and Engineering, Vol. 3, pp. 269-289, 1974.

[13] Sami Abdel-Fattah Abdel-Ghani El-Shaikh, "CFD Technique for Solving Low Water Level Problem of Axial Flow Pumps" American Journal of Water Science and Engineering Volume 3, Issue 3, May 2017, Pages: 34-44.