

# Flow Characteristics and Efficiency of Faraskour Pump Sump using Computational Fluid Dynamics (CFD)

Ashraf Ghanem\* and Elzahry Farouk Elzahry

Faculty of Engineering at Shoubra, Benha University, Egypt

\*Corresponding author: [ashraf\\_ghanem@outlook.com](mailto:ashraf_ghanem@outlook.com)

Submitted 14 June 2020; Revised 07 July 2020; Accepted 20 July 2020.

Copyright © 2020 The Authors.

**Abstract:** This paper predicts and enhances the hydraulic problems in the Faraskour pumping station. Initially, water could not reach the first and fifth units of the operation. The main hydraulic problem of the suction basin of the new pump station is the sharp rotation of the suction guide from the sharp rotation of the quay station, and that caused the continuous discontinuation of the first and fifth units due to the lack of regular water entering the unit. A numerical simulation was conducted to investigate the hydraulic stability of the station. Computational fluid dynamic (CFD) is used to simulate the flow conditions at different working pumping units to predict the hydraulic problem at the suction side. The results indicate that the geometry of the intake is proper for running five parallel flow pumps with the designed flow rate and use guide walls with a curvature length of 6 m and width of 0.5 m for each pump.

**Keywords:** CFD; Numerical simulation; Streamline pattern; Sump.

## 1. INTRODUCTION

During pump operations, abnormal flow phenomena such as cavitation, flow separation, pressure loss, vibration, and noise occur often by flow unsteadiness and instability. In many conditions, free and subsurface vortices containing air occurred in sump pumps seriously damage to the pump system. According to the JSME Standard, the appearance of such vortices is not permissible for ordinary pump sumps. Experiments with scale models have been performed to assess the performance of sump or to improve the design configuration of a sump [1]. This study attempts to model the flow characteristic in a pump sump of the physical model by using Computational Fluid Dynamics (CFD) code FLUENT. The experimental procedures include the data collection using a flow meter and a swirl meter (Rotometer /Vortimeter). Two types of measurements were conducted which are flow, and swirl angle. A visual test that involves the dye tracing technique was also carried out to characterize the flow. The CFD analysis is done at critical cases, grid generation is done in ICEM-CFD and numerical analysis is carried out in FLUENT. Water flow is analyzed with the help of velocity streamlines and vector plot and velocity contour at the entrance of pump chamber, in CFD-POST software.

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 [2]. The flow characteristics of pump sump and performance analysis of the mixed flow pump was conducted in [3]. The efficiency of anti-vortex devices to confirm the uniform flow. 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.

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. It 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 the design of sump geometry can be reduced to a large extent through CFD studies as we can predict the flow parameters at the pump inlet with the change in geometry without actual running of a pump with CFD. Hence the design of the sump can be optimized to keep the flow parameters below limiting values. Prakash *et al.* [4] attempted to minimize the swirl angles and increase the flow at the pump inlet.

Submerged vortex is introduced numerically by analyzing the flow in the pump sump with and without baffle plates [5]. 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 was considered in [6]. A design guideline for the shape of pump sump is suggested by model test and CFD analysis in [7]. Moreover, the Turbo-machinery Society of Japan [8] has revised the standard of pump sump model test and the numerical analysis can be possibly used 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 [9].

## 2. PROBLEM IDENTIFICATION AND METHODOLOGY

The study involves a new Faraskour drainage Pumping Station which is located at Faraskour drain Damietta governorate. It consists of five axial pumping large units with a discharge of  $5 \text{ m}^3/\text{s}$  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 at most four pumping units.

ANSYS Fluent 18.1 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 were examined for the operation of stopping units and operating of other units to reach the best scenarios for the operation of units within the station, and to identify and analyze the problem to control flow within the stream and entrances units. Figure 2(a) shows a two-dimensional image of the suction pump. The three-dimensional unstructured mesh (tetrahedral) has been created by using the structured mesh (hexagonal) as shown in Figure 2(b).

Meshing is a crucial part of the analysis in which the whole geometry is divided or broken down 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, 7221464 tetrahedral elements are used. The meshes have a total of 2657317 nodes. Nodes are the points on a meshed surface body where the different elements meet. Figure 2 show the surface, the volume mesh of the sump intake, and the geometry.

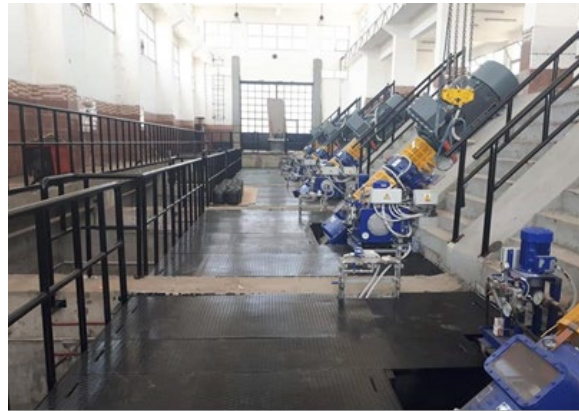


Figure 1. New Faraskour drainage pumping station

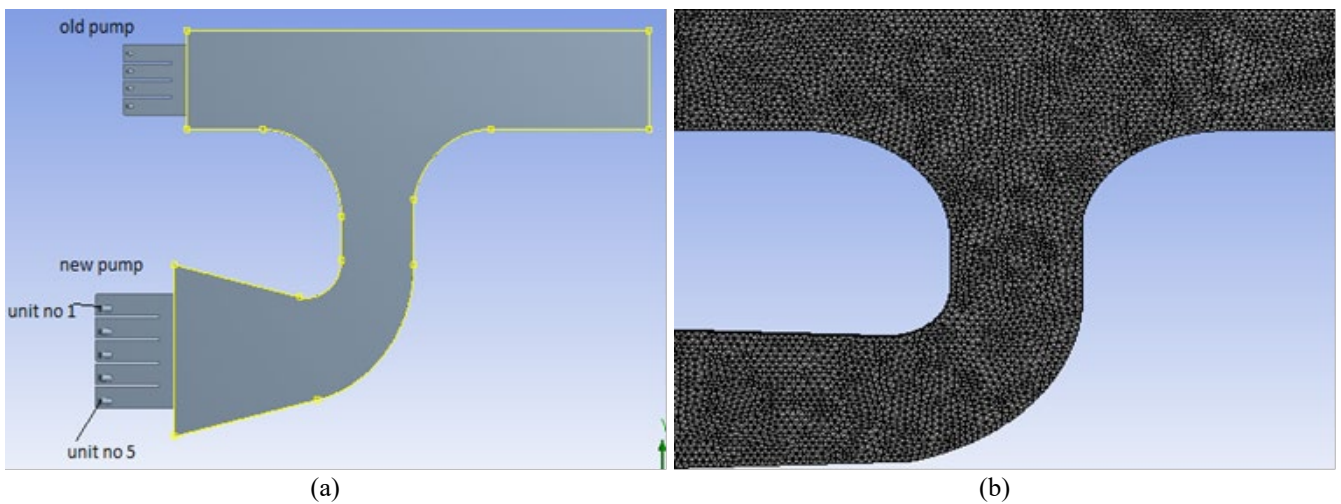


Figure 2. Two-dimensional image of the suction sump and unit input

### 3. BOUNDARY CONDITIONS AND GRID GENERATION

To get an accurate simulation, the outlet volume flow was applied in two cases. 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 ANSYS R18.1 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, and large-scale and time-dependent phenomena are considered directly.

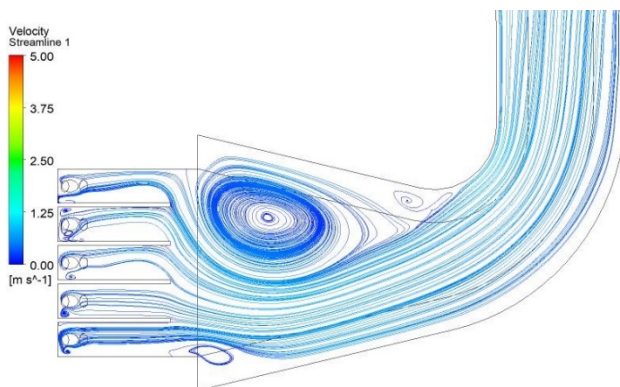
Three-dimensional unstructured meshes are used for the flow simulation in the pump sump. The unstructured mesh is used for this study due to model complexity and easy to mesh especially at the intake section. The numerical solver uses unstructured meshes that allow flexibility in meshing very complex geometries while maintaining high-quality computational mesh which is necessary for obtaining accurate solutions [10]. 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, 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.

### 4. EXISTING OPERATING CONDITION

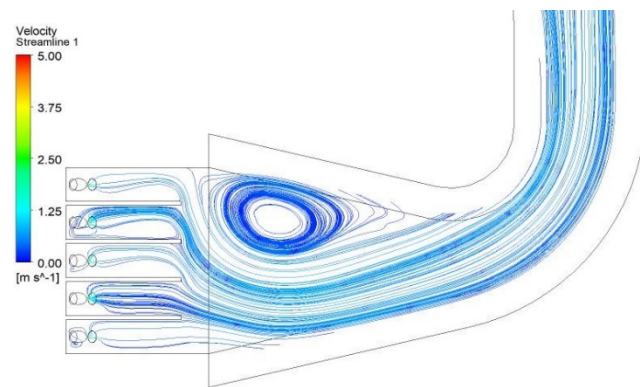
The existing operating condition contains four scenarios based on the shapes of the flow, its lines, and directions, as well as forms of speeds. Then the same four scenarios were examined after placement of routers (walls) close to the end of the suction channel to redistribute the lines flow within the stream at the entrance of units.

#### 4.1 Case 1: All Five Units Operating

For the first scenario, when all five units of the sump pump 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 4 and 5). In addition, it decreases with the internal bend of the suction path through Units 1, 2 and 3. Figure 3(a) shows the flow lines in the suction basin and the intake of the pumps. At the level of the bell, it is also 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). The figure 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. A map of velocities of the channel and the entrance shows a relatively static area (dead) with relatively low speeds towards Unit 1, as shown in Figure 3(c). On the other hand, from the shape of flow directions at the bottom level of the pipes of the pull, an effect of the swirl (whirlpools) and separation zones of the flow directions (especially Unit 1) can be noted 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 five units

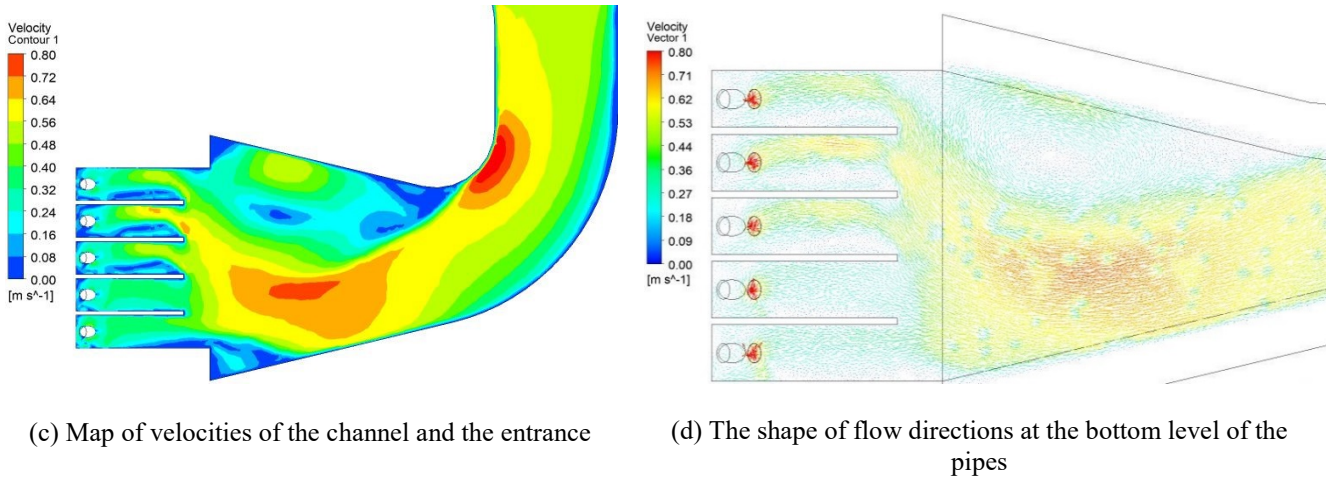


Figure 3. Case 1: All five units operating

**4.2 Case 2: Three Units Operating (3, 4 and 5)**

From the results of the first scenario, it is noted 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, another scenario was suggested by running the units separately to observe the flow distribution. In the second scenario when three units are operated (Units 3, 4 and 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 appeared at the entrance of Units 1 and 2, as shown in Figure 4(a) which shows the flow lines in the suction basin and the intake of the pumps. At the level of the bell, it also is noted that the direction of the flow lines is a regular distribution for Units 3, 4 and 5, and the swirls appeared at Units 1 and 2. Figure 4(b) shows the flow lines for different units in water level. However, a map of velocities shows that there is a drop zone of the velocity values on the inner bend of the intake, as shown in Figure 4(c). From the flow directions at the level of the bell, the presence of places of separation and swirls at Unit 1 was noted.

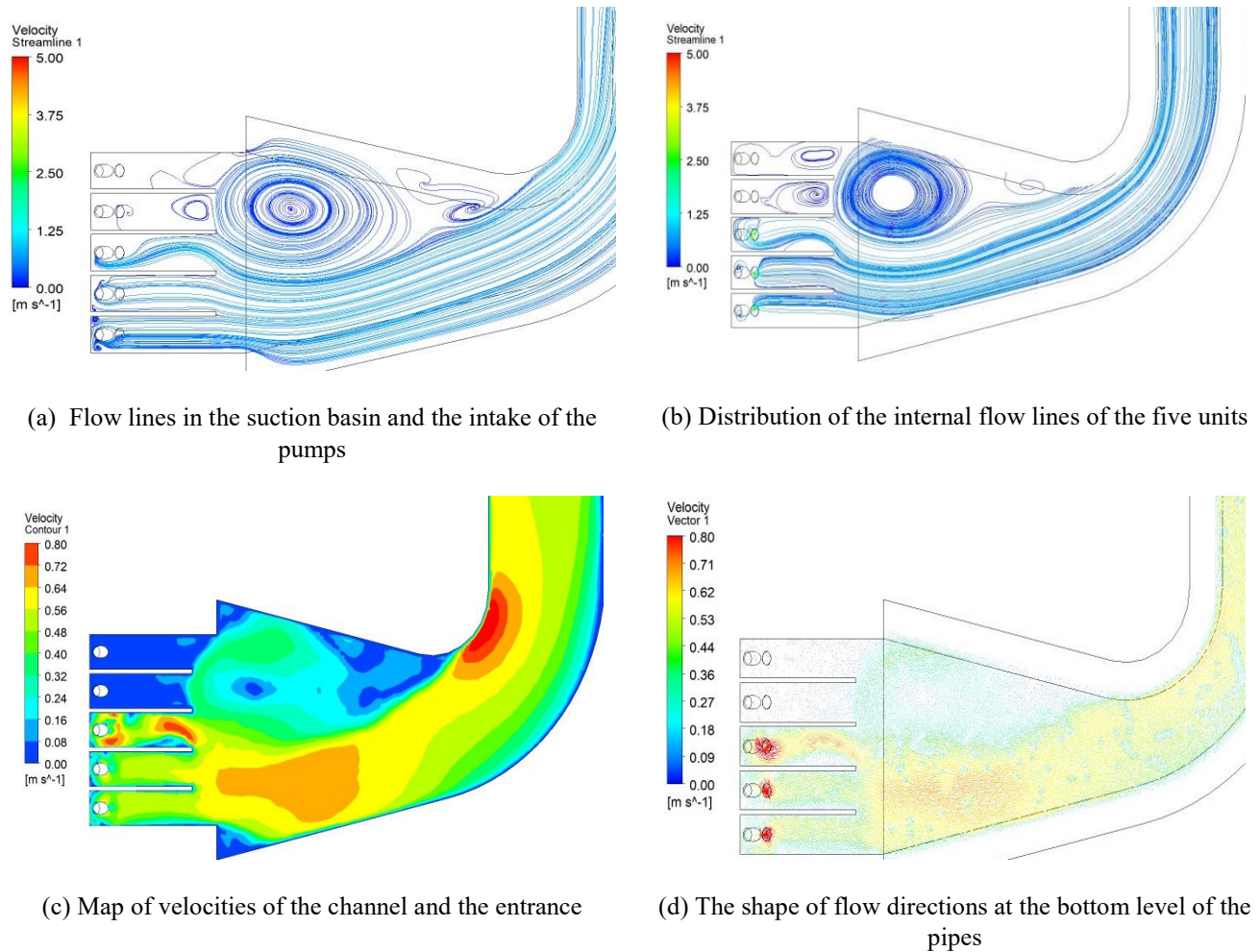


Figure 4. Case 2: Three units operating (Units 3, 4 and 5)

### 4.3 Case 3: Three Units Operating (1, 2 and 3)

In the third scenario when three units are operated (Units 1, 2 and 3 which located in the inner bend of the suction path), it is found that irregularity in the lines of flow at the maximum level and opposite direction for lines of flow far from Units 1, 2 and 3, the effect of the sharp curvature of the stream in the form of flow lines where it is disappeared at Units 1 and 2, as shown in Figure 5(a). At the level of the bell, the effect of turbulent flow by a sudden change in the direction of the flow lines before the intake of the working units and vortices in Unit 1 is clear, as shown in Figure 5(b). However, a map of velocities shows irregularities of the speeds at the intake of the three units, as shown in Figure 5(c). Finally, from the flow directions at the level of the bell, the presence of separation zones and vortices at the bell level for the three units is noted.

### 4.4 Case 4: Three Units Operating (1, 3 and 5)

In the fourth scenario when three units are operated (Units 1, 3 and 5), it is found that the flow lines were not distributed regularly on the bending and at the entrance of inputs unit and decreasing density of the flow lines in a unit, as shown in Figure 6(a). Also, flow lines of the bell level indicate irregularity of the incoming lines of the units, as shown in Figure 6(b). A map of velocities indicates that irregular velocities within the entrances of the working units is still exist, as shown in Figure 6(c). 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 Units 1 and 5.

### 4.5 Finding

From the results, two problems were noted: (1) 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 and 5 run together; (2) The presence of areas where velocities are relatively low which becomes places of accumulation of sediments, and affects the work of the units. Therefore, a redistribution of the flow lines in the suction basin and the entrance of the units were suggested to solve these problems. The solutions were the placement of routers or walls close to the bend of the suction channel at the end of the bend of directly at a distance of 20 m from the direction of the clouds in the direction of the entrance of the units and corresponding to the wall separating the units. This is to regulate and redistribute the flow lines at the entrance of units. The routers have a length of 6 m and width of 0.5 m, and located as shown in Figure 7.

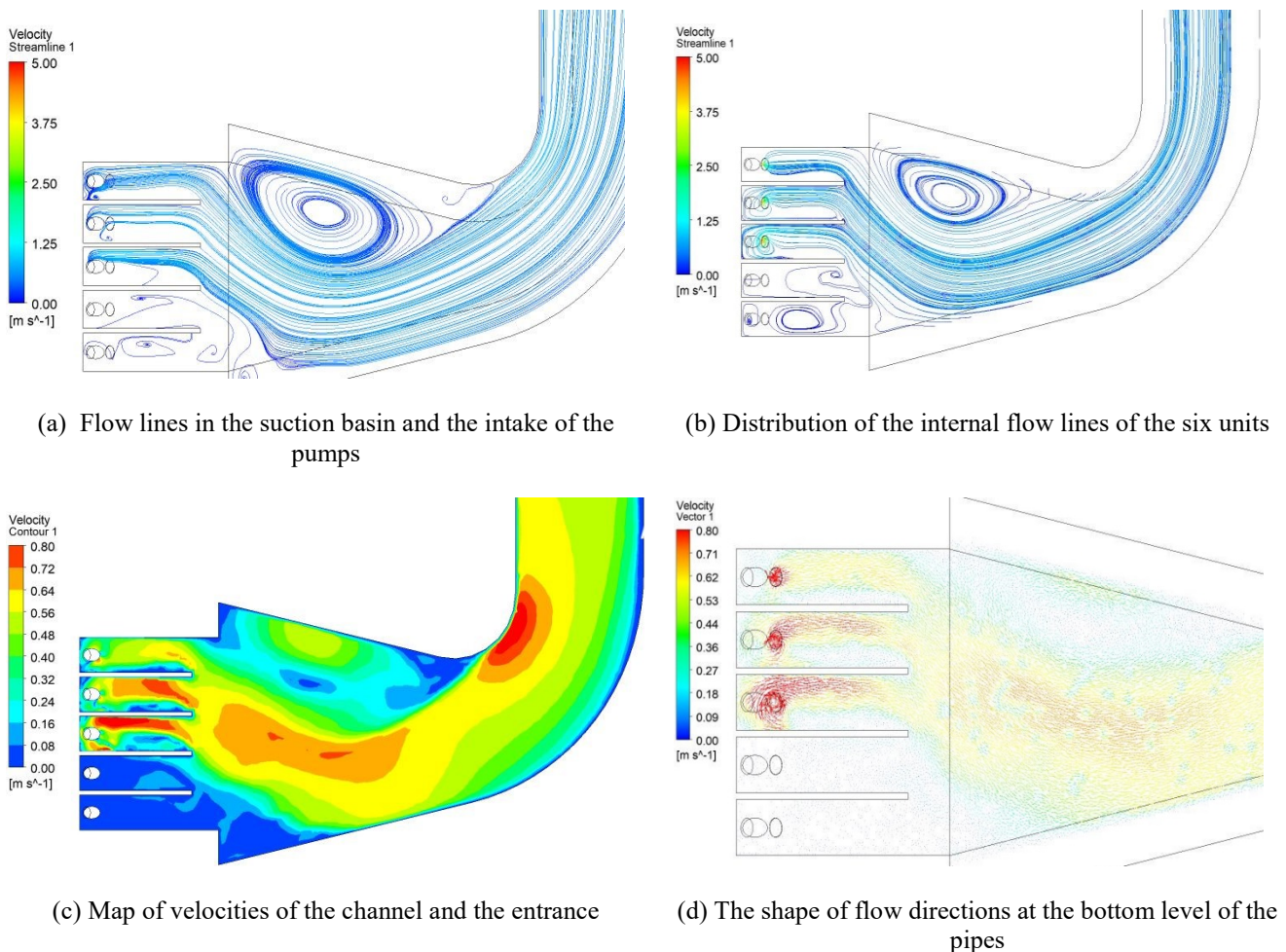


Figure 5. Case 3: Three units operating (1, 2 and 3)

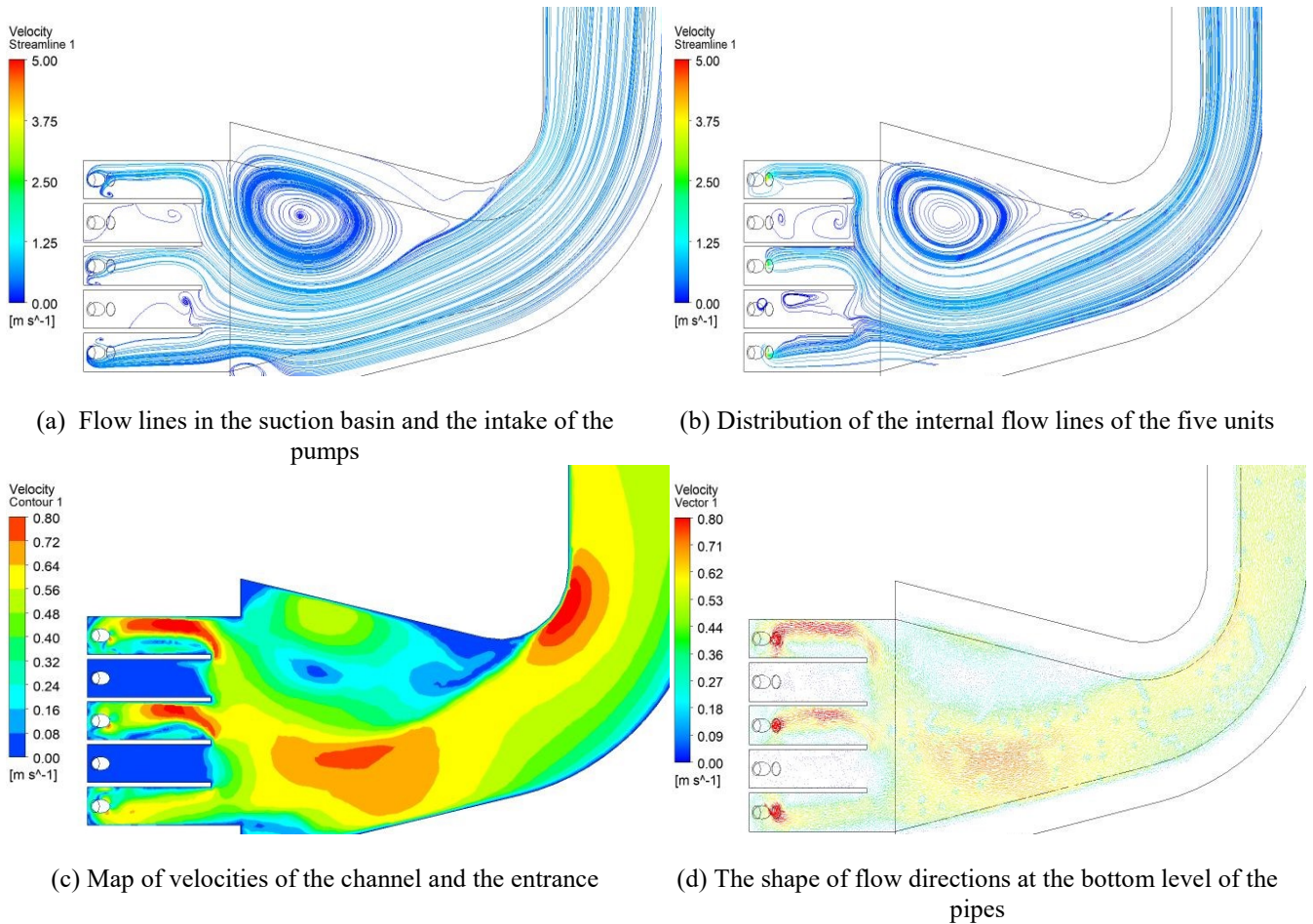


Figure 6. Case 4: Three units operating (1, 3 and 5)

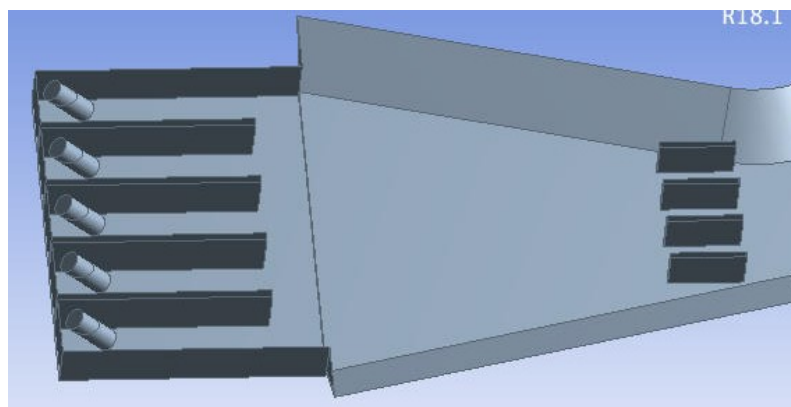


Figure 7. Setting the proposed routers through the suction sump

## 5. ADDING GUIDE WALLS

### 5.1 Case 1: All Five Units Operating

The results of the flow lines of the first scenario after adding guide walls show a good redistribution of the flow lines at the entrance of the units, especially at Units 1 and 2 which the flow lines as shown in Figure 8(a). Also, 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 Units 1 and 2 as shown in Figure 8(b). The speed map 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).

### 5.2 Case 2: Three Units Operating (3, 4 and 5)

The results of the flow lines of the second scenario after adding guide walls indicate the disappearance of the vortices the maximum level on the Units 3, 4 and 5, as shown in Figure 9(a), and disappearance of swirls at the level of the bell on the three units as shown in Figure 9(b). It is also clear that the guide walls worked on the uniformity of the velocity values to the

three working units but with a relatively short area of the relatively low speeds of the inner bend of the tube. However, it improved better than in the case of the same case without guide walls, as shown in Figure 9(c). Finally, the flow directions at the level of the bell show regularity as observed for Units 4 and 5, but there is a breakout area below Unit 5.

**5.3 Case 3: Three Units Operating (1, 2 and 3)**

The results of the flow lines of the third scenario after adding guide walls indicate good regularity of the flow lines at maximum water level, as shown in Figure 10(a), and uniformity of pulling for units below the bell as shown in Figure 10(b). The velocity values of the units are irregular as for the velocity map, as shown in Figure 10(c). As for the flow directions below the bell, it appears that the low effect of the swirl zones.

**5.2 Case 4: Three Units Operating (1, 3 and 5)**

The results of the flow lines of the fourth scenario after adding guide walls indicate that it becomes working uniformly, as shown in Figure 11(a). It is also appearing that a good distribution of the lines on all the units as shown in Figure 11(b), while Figure 11(c) shows a good distribution of speeds for working units.

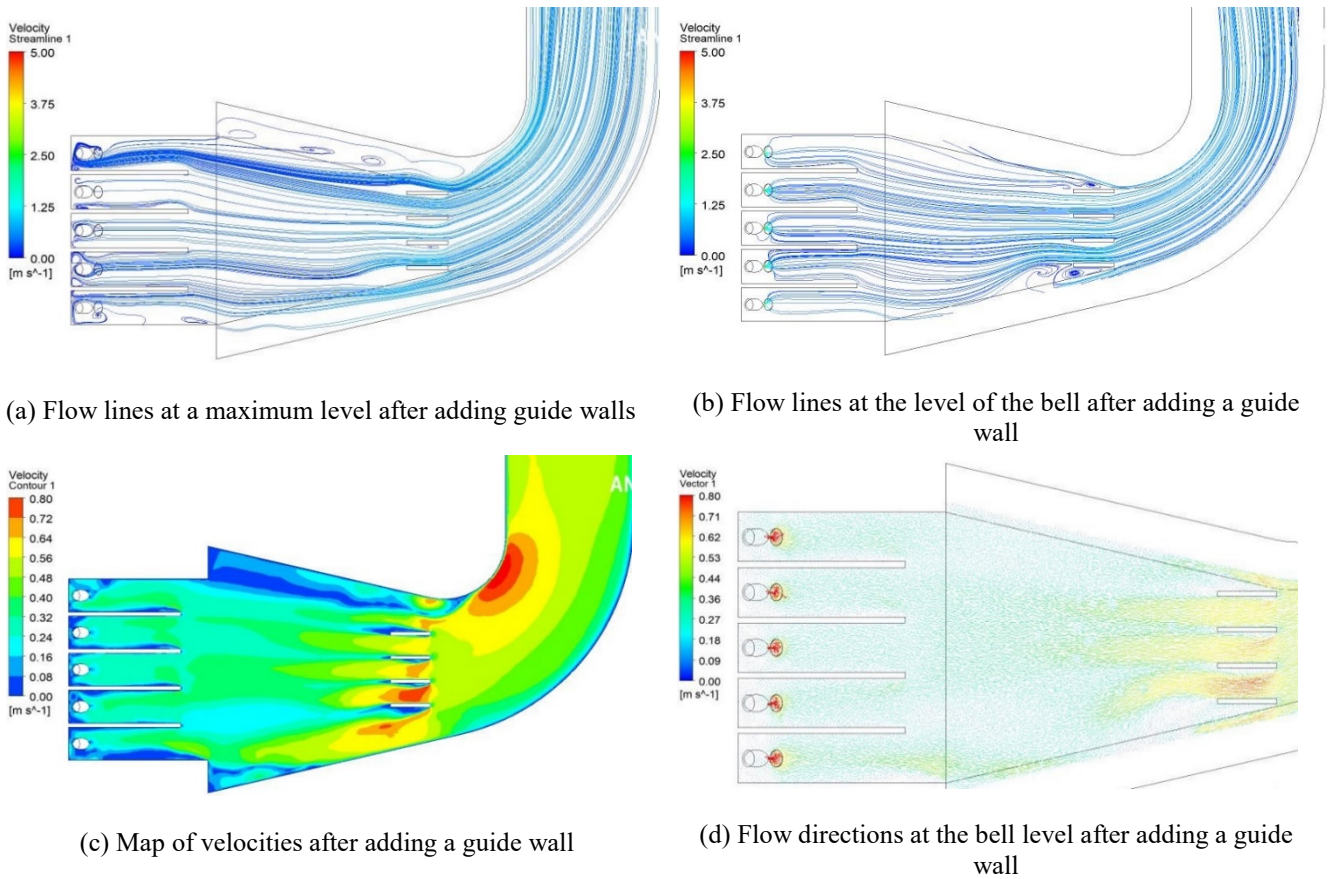
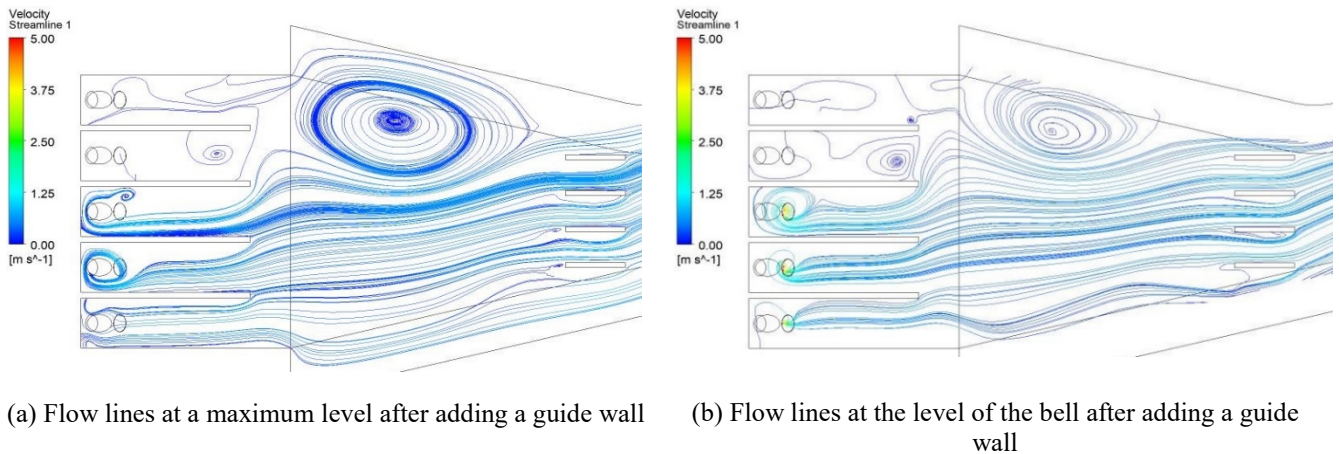
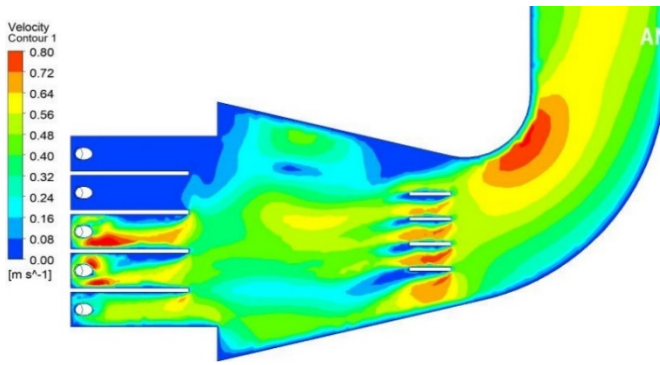
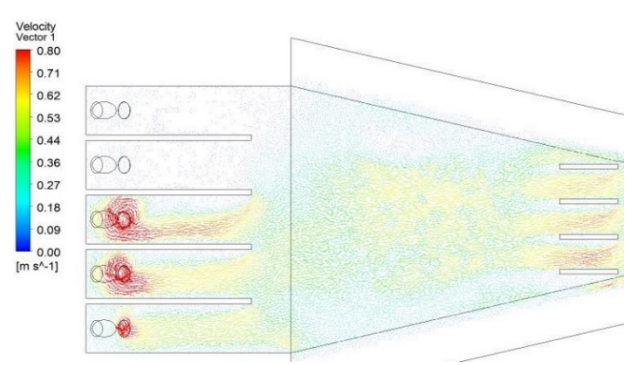


Figure 8. Case 1: All five units operating



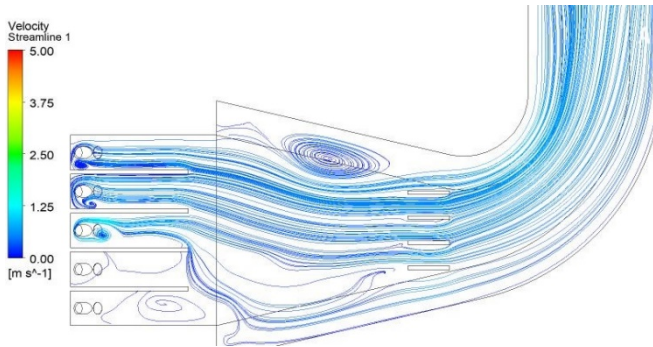


(c) Map of velocities after adding a guide wall

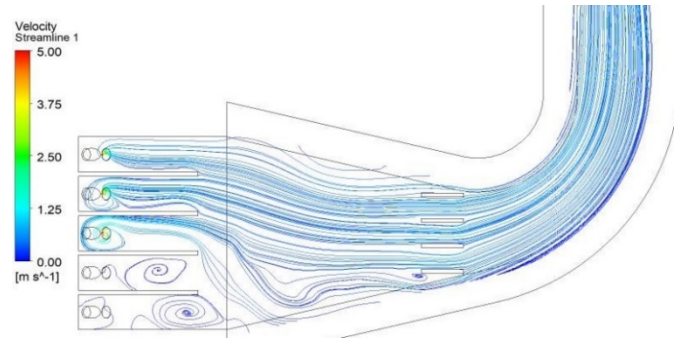


(d) Flow directions at the bell level after adding a guide wall

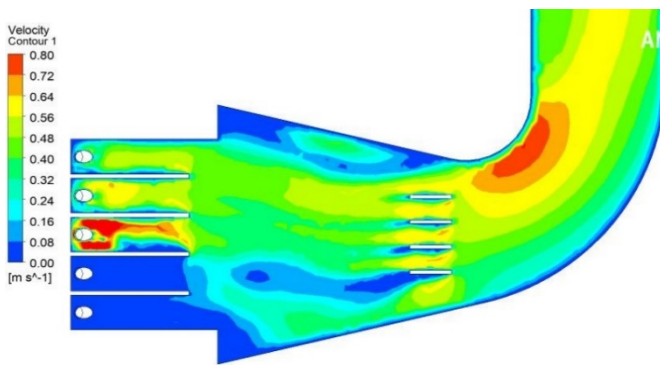
Figure 9. Case 2: Three units operating (3, 4 and 5)



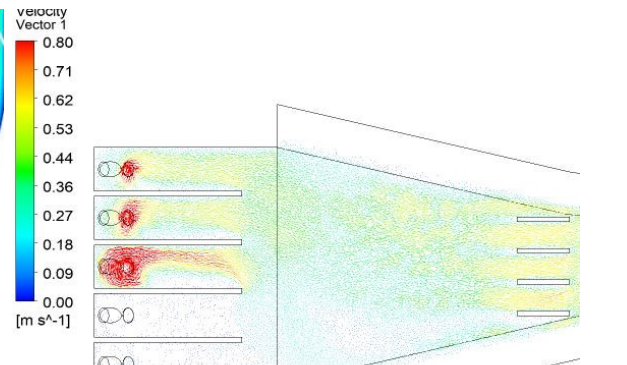
(a) Flow lines at a maximum level after adding a guide wall



(b) Flow lines at the level of the bell after adding a guide wall

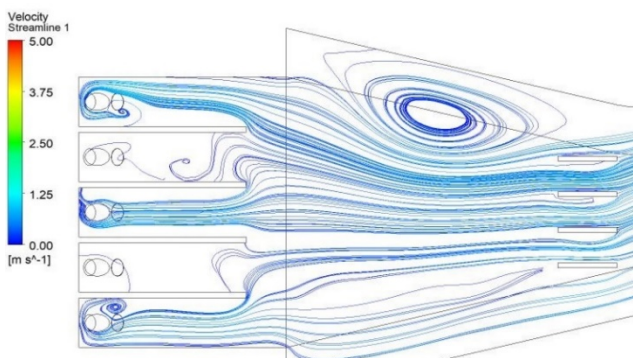


(c) Map of velocities after adding a guide wall

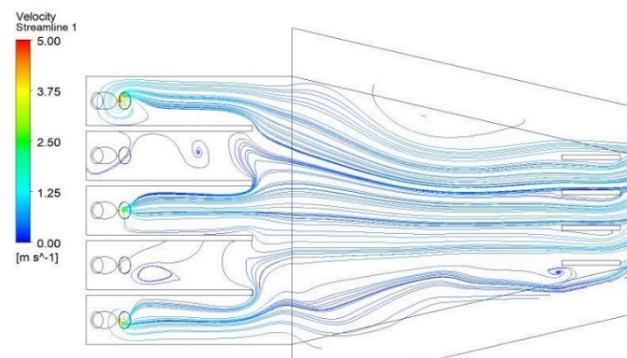


(d) Flow directions at the bell level after adding a guide wall

Figure 10. Case 3: Three units operating (1, 2 and 3)



(a) Flow lines at a maximum level after adding a guide wall



(b) Flow lines at the level of the bell after adding a guide wall

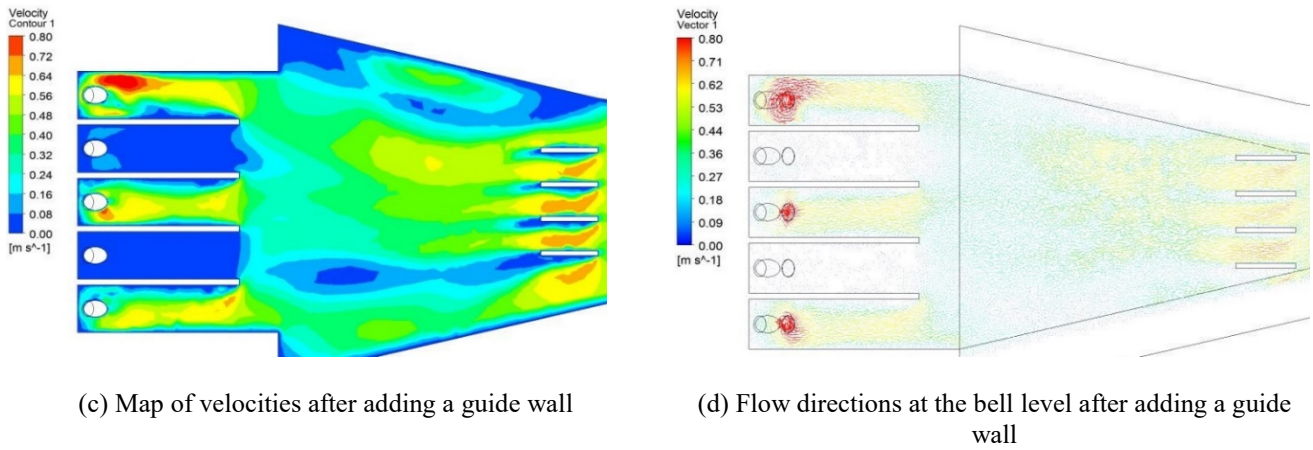


Figure 11. Case 4: Three units operating (1, 3 and 5)

## 6. CONCLUSIONS

It has been shown that the problems of flow in the new Faraskour station are due to the sharp bending in the intake at the entrance of the units to the station, bad distribution for the incoming flow lines of the units that leads to a low intensity at the internal bending especially at Units 1 and 2, and increase at the external bend at the Units 4 and 5. These problems affect the work of the units especially Units 1 and 5 when operates together. Two scenarios with four different cases were examined. The first scenarios were studied to determine the performance of the units under different circumstances, whereas the second scenario was studied after adding four guide walls. Simulation results show that the geometry of the intake is proper for running five parallel axial flow pumps with the designed flow rate and the proposed guide walls. It was shown that adding the guide walls improved the performance of the pumping station and the flow distribution at each unit.

## REFERENCES

- [1] Kadam Pratap M. and D.S. Chavan, CFD analysis of flow in pump sump to check suitability for better performance of pump, *International Journal on Mechanical Engineering and Robotics*, 1(2), 2013, 59-65.
- [2] S. M. Borghei and A. R. Kabiri-Samani, Effect of anti-vortex plates on critical submergence at a vertical intake, *Transactions A: Civil Engineering*, 17(2), 2010, 89-95.
- [3] Y. X. Zhao, C. G. Kim and Y. H. Lee, CFD study on flow characteristics of pump sump and performance analysis of the mixed flow pump, *Proceedings of 6th International Conference on Pumps and Fans with Compressors and Wind Turbines (ICPF)*, Beijing, 2013, 1-6.
- [4] Prakash Shinde, V. N. Raibhole and Ajinkya Satam, Numerical simulation for flow field in a sump, *Proceedings of 5th National Conference on Recent Developments in Mechanical Engineering*, Pune, 2016, 22-27.
- [5] 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, *Proceedings of the 9th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery (ISROMAC)*, Honolulu, Hawaii, 2002, 1-6.
- [6] 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), 2010, 1389-1400.
- [7] 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.
- [8] Turbomachinery Society of Japan, *Standard Method for Model Testing the Performance of a Pump Sump*, TSJ S002, 2005.
- [9] 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, *Proceedings of 26th IAHR Symposium on Hydraulic Machinery and Systems*, Beijing, 2012, 1-11.
- [10] B. E. Launder and D. B. Spalding, The numerical computation of turbulent flows, *Computer Methods in Applied Mechanics and Engineering*, 3, 1974, 269-289.