International Journal of Engineering Research and Technology.
ISSN 0974-3154 Volume 7, Number 1 (2014), pp. 79-91
© International Research Publication House
http://www.irphouse.com

Evaluation of the Flow Characteristics in the Intake Structure and Pump Sumps Using Physical Model

Essam A. Khalifa* Mostafa A. Abu-Zeid S. M. Abdel-Rahman*****
Sami A. A. El-Shaikh***

**Associate Professor, National Water Research Center,*
*** Chairman of the Mechanical & Electrical Department, MWRI, Egypt,*
abuzeidm@link.net
****Mechanical & Electrical Research Institute,*
National Water Research Center, Ministry of Water Resources

ABSTRACT

This paper presents a case study on the use of CFD modeling to re-evaluate and modify the design of a large plant influent pump station.
A sump is designed to provide adequate water supply to pumps installed in sump. It is also essential to design the sump to provide fairly uniform and swirl free flow to pumps. The Hydraulic Institute Standards specify general guidelines for the design of sumps and is based on extensive studies on variety of sumps [1]. In case the sump is designed as per the guidelines of HIS, it is ensured that the flow pattern will be relatively uniform and swirl free flow to pumps.
However it is not always possible to follow guidelines of HIS due to site constraints and to ensure uniform and swirl free flow to pump. Investigation of sump is required prior to installing the pumps at site.
In the present analysis, sump with a circular design is analyzed using latest computational analysis tools i.e., CFD tools. CFD study was carried out on initial sump geometry and initial CFD results were analyzed. Based on initial CFD results, final sump geometry is modified and its results are presented herein along with suggested geometrical modifications.

Key words: Intake, vortices, pump sump, flow velocity, Swirl, pre-rotation

INTRODUCTION

The basic purpose of a pump intake is to supply water with 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 [2]. It is essential to ensure that the pumps operating in such pump intakes get smooth swirl free flow at their inlets. Proper intake design provides 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 [3]. 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 smooth flow over the entire flow range of the pumps and in all the combinations of the pumps.

Traditional approach was to carry out sump model studies experimentally with a reduced scale model and applying Froude Similarity rules. Computational Fluid Dynamics (CFD) has recently come up as an alternative approach to investigate the complex fluid flow phenomenon in sumps [4]. It is rapidly becoming an important tool for analysis and design in hydraulic engineering. Hydraulic engineering encompasses a broad range of activity from flow in a river to design of structures to control and distribute/divert water for various purposes. These flow problems have features that are not commonly found in other applications. In the case of pump industry applications, CFD tools are important in view of analysis of the hydraulic passage of the pumps and sumps. For impeller inlet, CFD can help to improve the inlet flow distribution by proper designing / checking the quality of flow in pump sump [5]. In case of sumps, CFD analysis is used to investigate flow quality entering into the pump at various combinations of pumps in operation. The analysis is done at minimum water level with pumps running at duty point. The velocities in the sump are highest at minimum water level [6].

OBJECTIVE

Any flow pattern which departs from the one of the steady uniform flow is not desirable. Phenomena that cannot be present to an excessive degree are submerged vortices, free surface vortices, excessive pre-swirl of flow entering the pump chambers, non uniform spatial distribution of velocity at the impeller eye, excessive variations in velocity and swirl with time and entrained air/gas bubbles.It is prudent to investigate the flow pattern in entire sump i.e., pump chambers, fore-bay and pump bell etc before its actual construction. Undesirable flow condition like non-uniform flow, hydraulic jump, air entrainment, swirl beyond acceptable limits, submerged vortex etc. could be revealed right in advance of actual construction at site. Undesirable flow conditions could be eliminated / reduced to minimum, by incorporating the necessary modifications in sump and their effectiveness could be ascertained by studying the flow under modified conditions.

USE OF CFD

The development of computational tools has helped in resolving some of the issues in Sump Model studies [7]. The experimental study calls for enormous time and there are some inherent limitations of experimental activity as the exact modeling of

Reynolds Number, Froude Number & Weber number is not possible on geometrically similar smaller models. As the CFD analysis can be carried out on prototype also, the issues related to in-accurate prediction of prototype flows from model studies using smaller size experimental models do not come into picture. CFD tools avoid physical modeling and testing every time. Better and faster design of sump and its analysis leads to shorter design cycles. The civil construction work at site can progress faster based on CFD results. Due to various constraints the model sump has to destroy after the sump model study and again will have to make it, if required in future for solving sump problem. But the data generated in the CFD analysis of the sump can be kept for future reference [8].

CFD analysis can be used to see parts of the system or phenomena happening within the system that would not otherwise be visible through experimental analysis. CFD gives a means of visualizing and enhanced understanding of the fluid flow and hence better insight of the flow in sump [9, 10, 11, and 12].

CASE STUDY

In this sump water is supplied to the sump through the intake channels fig.1 (a) (b) shows the plan and 3D views of the sump geometry respectively. The pumping system consists of 3 nos. of BHMa 55 pumps with capacity of 4968 m3/hr for each pump, out of which 1 pump are in operation and other 2 are kept as a standby for one case. CFD analysis also carried out for all pumps are in operation.

It was proposed to carry out CFD analysis of the sump to see its overall suitability and to modify the geometry, in view of flow quality, if required. The CFD analysis of the sump includes a part of pump intake channel, forebay, intake bar screens and pipe representing the pump etc. The flow study is carried out for various combinations of pumps running at duty point at minimum water level (Please refer fig.1 for nomenclature of pumps).

The flow quality was not good for initial sump geometry. Several trails have been done to improve the flow quality. Results of the recommended geometry are presented herein. The flow study is done with under consideration that pumps working at minimum water level as the operation at minimum water level is considered to be worst.

CFD ANALYSIS TOOLS

CFD tools are currently used worldwide to get advance information on hydraulic performance of systems. For this analysis the latest version of ANSYS CFX 14.0 is used.

The surface model of the sump geometry of is created using well known solid modeler Pro-Engineer Wildfire (Creo-1). The geometry (.iges form) is taken into ANSYS ICEM-CFD 14.0 software for good quality grid generation. This grid file is further taken into ANSYS CFX-Pre 14.0 for applying the suitable physics to the geometry. CFX-Solver and CFX-Post is used to solve and analyzing the results respectively.

BOUNDARY CONDITIONS

In CFD analysis, solution depends upon the appropriate boundary conditions. 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 into the sump. The boundary conditions at the outlets were specified in terms of mass flow / pressure boundary i.e., amongst all outlets any one of outlet is specified in terms of pressure boundary and remaining outlets is specified in terms of mass flow boundary. The free surface was treated as a surface with zero normal gradients of flow properties. This is applied in terms of symmetry boundary condition at the top. All the pipes representing the Pumps are thin surfaces and treated as walls with no slip. Since this sump is located in sea, hence side walls of inlet domain are considered walls with free slip.The outer circular surface of sump is considered as wall with no slip. The domain type is fluid domain with water as the flowing fluid in the domain. The standard κ-ε turbulence model is used for analysis. This is one of the most prominent turbulence models. The κ-ε model has been implemented in most general purpose CFD codes and is considered the industry standard model.

It has proven to be stable and numerically robust. For general purpose simulations, the κ-ε model offers a good compromise in terms of accuracy and robustness. The turbulence model uses the scalable wall-function approach to improve robustness and accuracy when the near-wall mesh is very fine. The scalable wall functions allow solution on arbitrarily fine near wall grids, which is a significant improvement over standard wall functions.

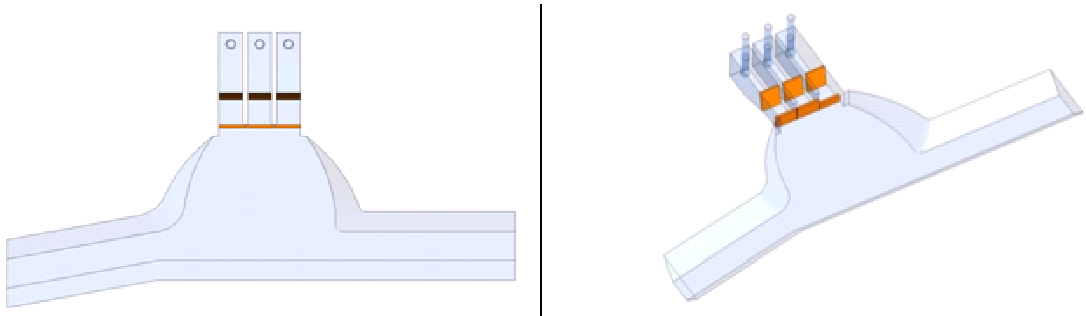
SOLVER SCHEME USED FOR SOLVER

The advection scheme that is used is 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 10-5 RMS level. The convergence of Mass and momentum is ensured in the solution.

RESULTS & DISCUSSION

CFD RESULTS OF INITIAL GEOMETRY

The initial geometry is prepared as per drawings and given in Fig 1. The initial geometry is prepared as per drawings and given in Fig 1. Fig.1 (a) shows the plan view and fig.1 (b) shows the 3D view of the sump geometry. For the analysis some part of the common canal is consider. Pump house is nearly perpendicular to the common canal. In the original geometry there is a forebay in between common canal and the pump chambers. Flow direction is shown in the fig.1 (a).In the original sump geometry there was a blocking wall of unequal height also consider for the analysis to see the effect of the wall. There is a screen at 4.5 m from the pump centerline. There are 3 pumps in this pump house which are modeled as a pipe with bell and column pipe.

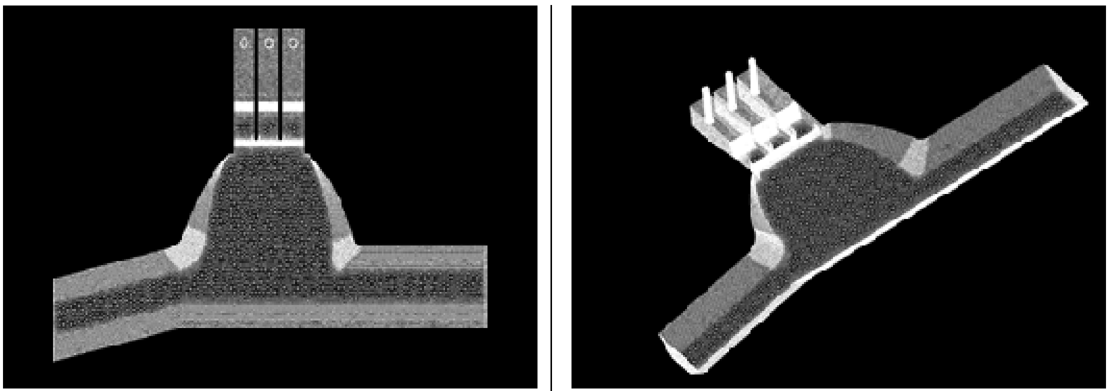


(a) (b)

Fig.1. Initial Sump Geometry used for CFD analysis; (a) Plan View (b) 3D View

The corresponding grid structure for the initial sump geometry is shown in Fig 2. The generated mesh is unstructured hybrid tetrahedral mesh with

Mostly tetrahedral elements.Total nos. of nodes is approx.0.4 million in the sump. Fine tetra elements were used near the wall pump chambers to achieve good computational results.



(a) (b)

Fig.2. Surface Grid Plot of Initial Sump; (a) Plan View (b) 3D View

COMBINATION OF TEST CASES

- **Working X Stand-By**

Table (1)-The combination of pumps in operation & pumps kept in standby mode for test cases is as per.

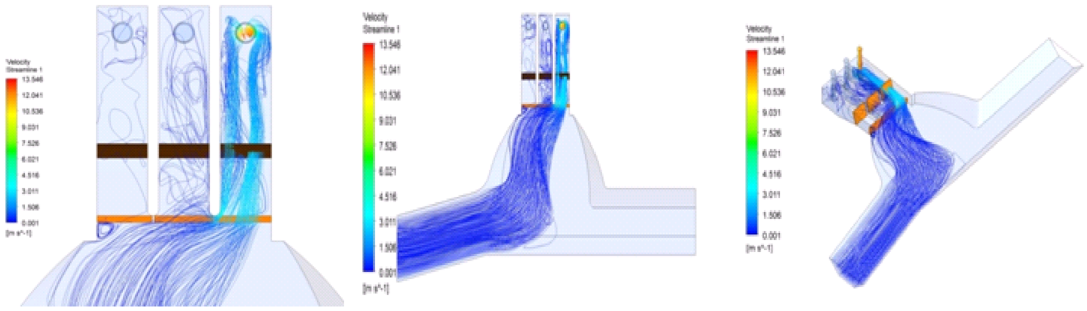
Pump	P-1	P-2	P-3
Original-1		*X*	*X*
Original-2	X		*X*
Original-3	*X*	*X*	
Original-4			

Modified-1		*X*	*X*
Modified-2	X		*X*
Modified-3	*X*	*X*	
Modified-4			

CFD RESULTS OF INITIAL SUMP GEOMETRY

The qualitative results of CFD analysis of initial geometry for all combinations of test cases are presented in terms of streamline plots (Ref Fig 3.1 & 3.4).

ORIGINAL-1 (P-1Working)



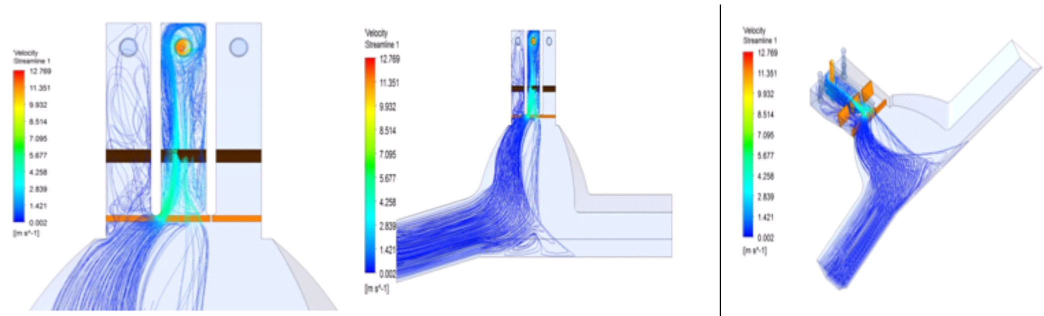
(a) **(b)**

Fig.3.1. Case 1 Pump 1 operate with Swirl Angles 17.84^0; (a) Plan View (b) 3D View

It is observed from the streamline plots of the sump geometry in fig.no.3.1 when P-1 is in operation and other pumps are in standby mode. It is observed that the flow is not getting distributed uniformly in the forebay. Large amount of recirculation zone and non-uniform flow in the pump chamber this may add to formation of swirl while flow enters in the bell. Due to this, flow pattern is almost un-symmetric in entire pump chamber. The swirl angle is found for this test case is 17.84° which is much higher than the acceptable limit of 5° (As suggested in HIS).

Hence sump geometry is not adequate for safe operation of pumps and sump. Thus for efficient and safe operation of pumps and sump, it is required to modify the sump geometry.

ORIGINAL-2 (P-2 Working)

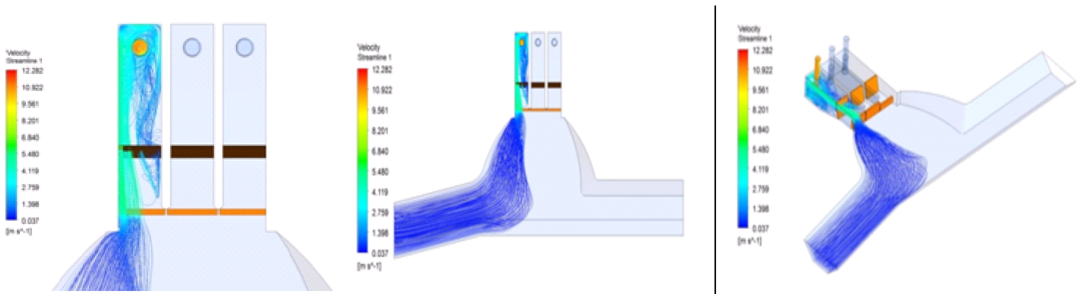


(a) **(b)**

Fig. 3.2 Streamline plot for Case 2 Pump 2 operate with Swirl Angles 35.81^0; (a) Plan View (b) 3D View

It is observed from the streamline plots of the sump geometry in fig.no.3.2 when P-2 is in operation and other pumps are in standby mode. It is observed that the flow is not getting distributed uniformly in the forebay. Large amount of recirculation zone and non-uniform flow in the pump chamber this may add to formation of swirl while flow enters in the bell. The swirl angle is found for this test case is 35.81° which is much higher than the acceptable limit of 5° (As suggested in HIS).

ORIGINAL-3 (P-3 Working)

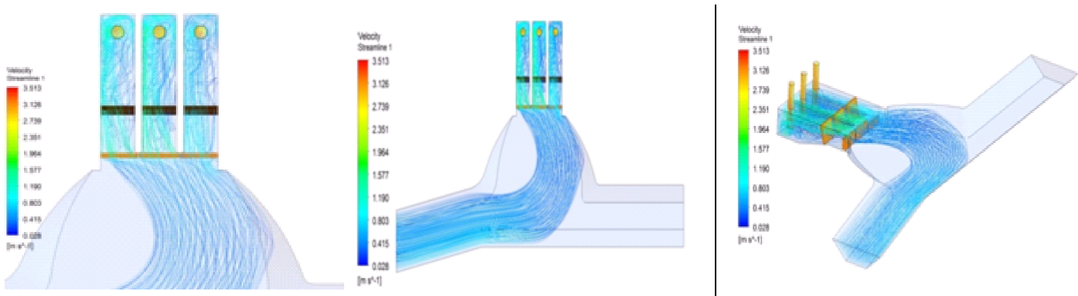


(a) **(b)**

Fig. 3.3 Streamline plot for Case 3 Pump 3 operate with Swirl Angles 53.84^0; (a) Plan View (b) 3D View

It is observed from the streamline plots of the sump geometry in fig.no.3.3 when P-3 is in operation and other pumps are in standby mode. It is observed that the flow is not getting distributed uniformly in the forebay. Large amount of recirculation zone and non-uniform flow in the pump chamber and especially behind and around the pump this may add to formation of swirl while flow enters in the bell. The swirl angle found is maximum 53.84° for this case. Which is much higher than the acceptable limit of 5° (As suggested in HIS)?

ORIGINAL-4 (All Working)



(a) (b)

Fig. 3.4 Streamline plot for Case 4 All Pumps operate with Swirl Angles 10.360, 21.430, and 15.230 respectively; (a) Plan View (b) 3D View

It is observed from the streamline plots of the sump geometry in fig.no.3.4 when all pumps are in operation. It is observed that flow is not attached to the sidewalls of the forebay. Non-uniform flow recirculation zone and in the pump chamber and especially behind and around the pump this may add to formation of swirl while flow enters in the bell. The maximum swirl angle is found 21.43^{0}° for this case. Which is much higher than the acceptable limit of 5° (As suggested in HIS)?

As per the CFD results of the initial geometry for all test cases the quantitative results are given in the form of swirl angles in Table-2 it is found that the sump geometry is not adequate for safe operation of pumps and sump. Thus for efficient and safe operation of pumps and sump, it is required to modify the sump geometry.

Table (2)-Swirl angle values for test cases of Initial sump geometry.

Pump	P-1	P-2	P-3
Original-1	17.84^{0}	X	X
Original-2	X	35.81^{0}	X
Original-3	X	X	53.89^{0}
Original-4	10.36^{0}	21.43^{0}	15.23^{0}

MODIFICATIONS IN INITIAL SUMP GEOMETRY

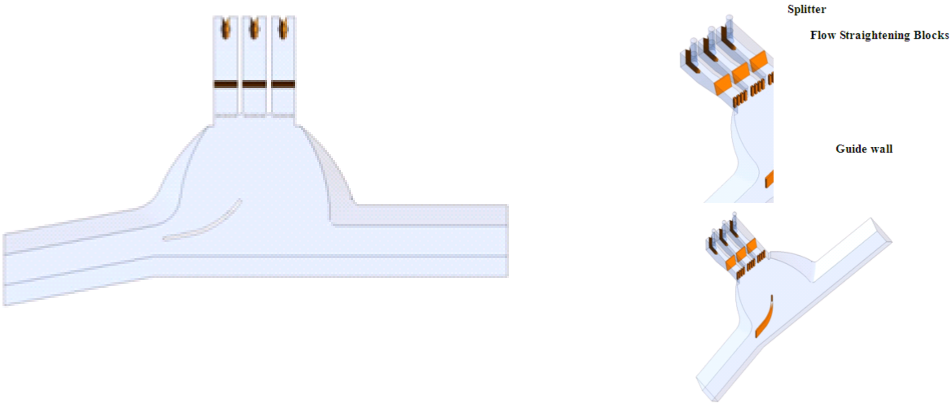
CFD analysis of initial sump geometry was carried out and flow pattern is not found adequate. To have uniform flow entry in forebay and to reduce the recirculation zone in front of the corner pump chambers following geometrical modifications are done in the initial sump geometry for efficient and safe operation of pumps:-

1) Guide walls are placed at common channel to make the flow distribution uniform in all pump chambers.
2) Array of Flow Straightening Blocks is placed in the forebay to make the uniform flow distribution in the pump chamber.

3) Floor and back wall splitters are added inside the pump chamber to avoid the recirculation of the flow behind and near the pumps.

MODIFIED GEOMETRY

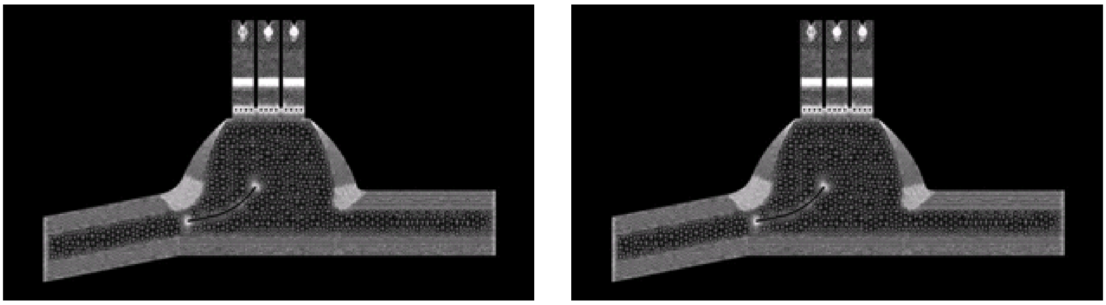
The modified sump geometry is given in Fig 4. The corresponding grid structure for the modified sump geometry is shown in Fig 5.



(a) **(b)**

Fig.4. Modified Sump Geometry for CFD analysis; Plan View (b) 3D View

MODIFIED GEOMETRY MESH



(a) (b)

Fig.5.Surface Grid Plot of Modified Sump Geometry; (a) Plan View (b) 3D View

CFD RESULTS OF MODIFIED SUMP GEOMETRY

CFD analysis has been carried out for modified sump geometry. The qualitative results of CFD analysis of modified sump geometry are presented in terms of streamline plots (Fig 6.1 to 6.4).It is observed from the streamline plots of the sump geometry that flow pattern is almost uniformly distributed in entire pump chamber.

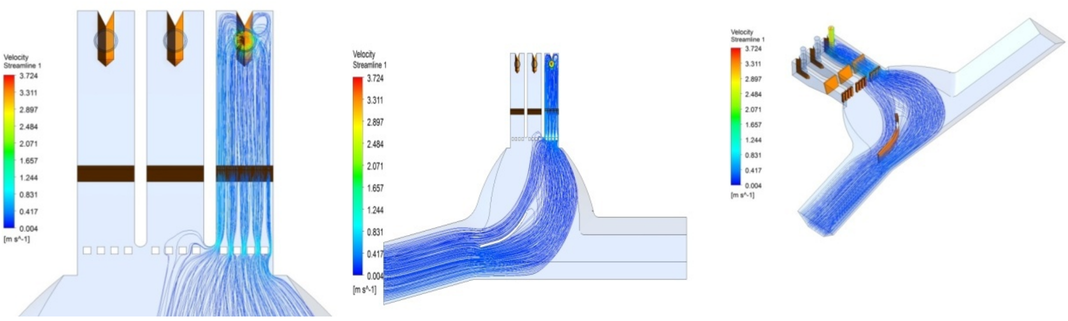
The modifications which are provided, ensures an equal distribution of the flow into the pump chamber. The recirculation zone is reduced to significant level and flow is entering uniformly into the suction bell of the pump. Some low velocity regions without any vortices are seen in front of the stand by pump chamber, which does not have significant contribution to the main flow. Also the strength of vortices formation at appropriate height of the pump is very low.

The quantitative results are given in the form of swirl angles in Table-3. Swirl angles are measured at the appropriate locations in the pump. The table indicates that swirl angles are within acceptable limit acceptable limit of 5° (As suggested in HIS) for efficient operation of pumps/sump.

Table (2)-Swirl angle values for test cases of Initial sump geometry.

Pump	P-1	P-2	P-3
Modified-1	2.29^0	X	X
Modified-2	X	0.25^0	X
Modified-3	X	X	0.50^0
Modified-4	1.92^0	1.42^0	1.80^0

MODIFIED-1 (P-1 Working)

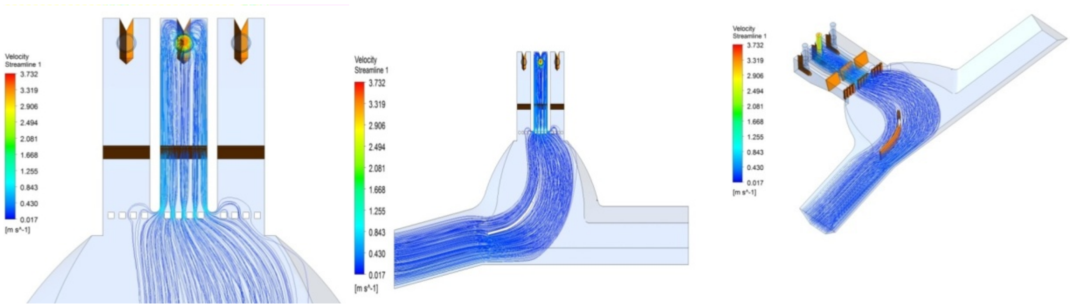


(a) **(b)**

Fig.6.1. Streamline plot for Case 5 modified Case 1 with Swirl Angles 2.29^0; (a) Plan View (b) 3D View

It is observed from the streamline plots of the sump geometry in fig.no.6.1 when P-1 is in operation and other pumps are in standby mode that flow pattern is almost uniformly distributed in entire pump chamber. The modifications which are provided, ensures an equal distribution of the flow into the pump chamber. The recirculation zone is reduced to significant level and flow is entering uniformly into the suction bell of the pump. The swirl angle found is 2.29° for this case which is within the acceptable limit of 5° (As suggested in HIS).

MODIFIED-2 (P-2 Working)

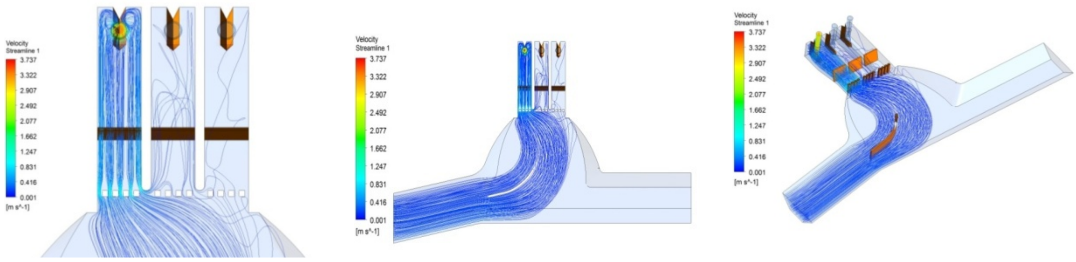


(a) (b)

Fig.6.2. Streamline plot for Case 6 modified Case 2 with Swirl Angles 0.25^0; (a) Plan View (b) 3D View

It is observed from the streamline plots of the sump geometry in fig.no.6.2 when P-2 is in operation and other pumps are in standby mode that flow pattern is almost uniformly distributed in entire pump chamber. The modifications which are provided, ensures an equal distribution of the flow into the pump chamber. The recirculation zone is reduced to significant level and flow is entering uniformly into the suction bell of the pump. The swirl angle found is 0.25° for this case which is within the acceptable limit of 5° (As suggested in HIS).

MODIFIED-3 (P-3 Working)

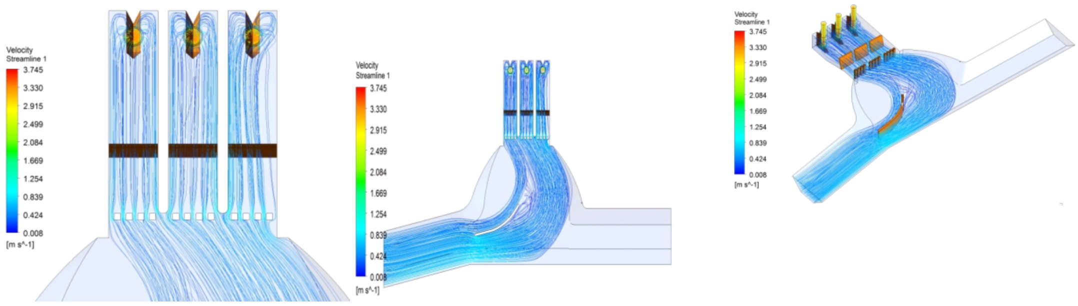


(a) (b)

Fig.6.3. Streamline plot for Case 7 modified Case 3 with Swirl Angles 0.25^0; (a) Plan View (b) 3D View

It is observed from the streamline plots of the sump geometry in fig.no.6.2 when P-2 is in operation and other pumps are in standby mode that flow pattern is almost uniformly distributed in entire pump chamber. The modifications which are provided, ensures an equal distribution of the flow into the pump chamber. The recirculation zone is reduced to significant level and flow is entering uniformly into the suction bell of the pump. The swirl angle found is 0.25° for this case which is within the acceptable limit of 5° (As suggested in HIS).

MODIFIED-4 (All Working)



(a) (b)

Fig.6.4. Streamline plot for Case 8 modified Case 4 with Swirl Angles 0.25^0; (a) Plan View (b) 3D View

CONCLUSION

The Computational Fluid dynamics (CFD) analysis of the sump is carried out to confirm check the suitability of the sump. It is observed from the streamline plots of the sump geometry that flow pattern is almost uniformly distributed in entire pump chambers due to appropriate geometrical modifications. Swirl angle is also within acceptable limit of 5° (As suggested in HIS).

Geometrical modifications suggested are:-

- Guide wall in the forebay
- Array of flow straightner blocks
- Floor and back wall splitters

Hence proposed sump geometry is recommended with the suggested geometrical modifications.

REFERENCES

[1] American National Standard for Pump Intake design (ANSI / HI 9.8-1998) by Hydraulic Institute, 9 Sylvan Way, Parsippany, New Jersey, USA.

[2] Shazy A. Shabayek1, *"Improving Approach Flow Hydraulics at Pump Intakes"* International Journal of Civil & Environmental Engineering IJCEE-IJENS Vol: 10 No: 06, IJCEE, Nov. 10, 2010.

[3] 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, c Sharif University of Technology, April 2010.

[4] Shyam N. shukla, and J. T. Kshirsagar, "Numerical prediction of air entrainment in pump intakes", Proceedings of the Twenty-Fourth International Pump Users Symposium, 2008.

[5] Khalifa, E.A. "Sensitivity Analysis of a Groundwater Flow Model" ASCE,

regional conference on environmental impacts on civil engineering technology. Cairo, Egypt, October 10-12, 1995.

[6] Cecilia Lucino, Sergio Liscia y Gonzalo Duró, "Vortex Detection in Pump Sumps by Means of CFD", Xxiv Latin American Congress on Hydraulics Punta Del Este, Uruguay, (Iahr), November 2010.

[7] ANSYS-CFX users guide, V. 14.0, By M/s ANSYS Inc., USA.

[8] Tanweer S. Desmukh & V.K Gahlot, *"Simulation of Flow through A Pump Sump and its Validation",* IJRRAS 4 (1), July 2010.

[9] Sami A. A. El-Shaikh, “Enhancing Hydraulic Performance of Pump Intakes Using Computational Fluid Dynamics (CFD): Case Study”, Water Science Journal, 2014

[10] Khalifa, E.A. "Groundwater Mathematical Model" is published in U.A.E. Japanese Workshop on "Water Resources and Greening in Desert" Abu Dhabi, January 28-29, 1995.

[11] Abu-Zeid, M.A., ” The effect of Open Sump Intake on the Pump Performance, ” 11th ICID International Drainage Workshop organized by Egyptian National Committee on Irrigation & Drainage (ENCID) & National Water Research Center (NWRC), 2012.

[12] Abu-Zeid, M.A, ” Finite Element Analysis For Prediction Hydraulic Performance Of A Rectangular Flap Valve, ” International Journal of Engineering Research & Technology (IJERT)-International Research Publication House, 2013.