ISSN 2395-1621

Experimental optimisation of vertical pump intake structure hydraulic parameters

[#1]P.D. Shinde,[#2] S.Y. Bhosale

[1]Prakashshinde_army@yahoo.com
[2]sybmodern@gmail.com

[#1]PG Scholar, Department of Mechanical Engineering, Savitribai Phule Pune University
PES Modern COE, Pune, India

[#2]Faculty, Department of Mechanical Engineering, Savitribai Phule Pune University
PES Modern COE, Pune, India

ABSTRACT

This Operational stability and smooth performance of vertical centrifugal pumps are greatly depending upon the entire pump, piping system and suction hydraulic conditions. Generally it is observed that pump set operated satisfactorily in the lab / factory while results different kind of the problem in the actual site / at the field of pumping system installation. Intake hydraulic geometry parameters results in major change in pump suction condition. Therefore sump model study is necessary to examine the flow structure in the intake. In experiment these parameters are maintained by optimized geometry, installing flow guiding devices in the sump, as a curtain wall, maintaining the proper floor angles. This study is concerned with experimental and computational investigation of the fluid parameters in the intake sump structure of the pump. The intake geometry parameters were, chamber width 2400 mm, length is 7025 mm, curtain wall of thickness 500 mm and bottom clearance of 2600 mm was placed at distance of 5000 mm from pump centre. Triangular type vortex breaker was added below the pump at height of 500 mm and length of 1875 mm. The optimized geometry shows similarity in the result of experimental sump model study and computational fluid analysis. In result, maximum swirl angle found was 4° with uniform flow. Intermittent Surface vortices (Type-1) were observed at upstream of curtain wall in forebay, but were not entering in to pump chamber. The slight surface recirculation zone was observed in the forebay but same was not entering in the pump chamber because of the curtain wall. The experimental sump model was scaled down by a ratio of 1:8 whereas the CFD sump model was scaled to the actual size.

Keywords— **computational fluid dynamics, entrained air, pump intake hydraulic parameters, scaled model study, submerged vortices, swirl angle.**

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

I. Introduction

Vortex formation around the pump intake in the pump sump makes the deterioration of pump and pumping system performance. Existence of the strong vortex in the water, the vortex causes swirl flow around the pump intake and results in the decrease of pump efficiency. Any pumping system mainly consists of the intake structure, main pumping unit and discharge system. The suction system largely consist of fluid (mostly water) canal, intake channel, forebay and sumps / well for the pump along with electromechanical operated devices like filter screen, gates, silt removing arrangement etc. Generally it is observed that pump set operated satisfactorily in the lab / factory while results different kind of the problem in the actual site / field of pumping system installation. In this context pump is mostly affected by the improper pump suction conditions which results in the affected hydraulic parameters for the satisfactory operation of the pumping unit. The free discharge of the fluid generates turbulence in the suction side as well as air entrains in the fluid. Entrained air, high velocity causes vibration, noise, shaft breakage, loss of prime and accelerates the corrosion of the pump suction parts.

So there are several design considerations in the design of the intake structure for the vertical Pumps intake design in early days done by physical modeling of the structure with maintaining the similarity. In this proper model scaling is required for sump design. For prediction of hydraulic parameters numerical simulation and programming with modifications in the flow geometry is required. There are 4 pumps installed in the sump, out of which 3 nos. pumps are working and 1 no. of pump is stand by. The duty point of the pump is 7500 M3/hrs at 22 M head. This study is aimed to examine the physical as well as computational fluid dynamics methods to analyze the flow behavior. A geometrically and dynamically scaled model is used for the physical model study and ANSYS Fluent is used for the numerical simulation. The flow pattern, turbulence, air entrainment, surface and sub surface vortices and flow velocity is analyzed. This study is aimed to examine the flow characteristics in the pump sump, suppression of the submerged vortex by using anti vortex apparatus, which occurs around the sump, by experimental and numerical methods to optimize the sump geometry.

I. EXPERIMENTAL AND NUMERICAL METHOD

The task of the numerical analysis and physical sump model study is to ascertain the suitability of the sump for the safe pump operations of various combinations of pumps.

A. Experimental Setup

The sump model rig includes geometrically similar hydraulic passages from part of inlet pipe, the forebay and pump chambers and upto pump suction bell as relevant to sump model test. Simulation to prototype sump will be achieved by selecting a scale ratio as 1:8. The model sump is constructed from M.S. sheets / wooden board and transparent acrylic sheets wherever needed. The back wall portion of sump will be made from transparent acrylic sheets. The model pump consists of transparent acrylic simulated suction bells connected to transparent acrylic pipe simulating the internal diameter of column pipe of prototype pump. Swirl meter will be placed at suitable location. The speed of swirl meter speed taken as measure of extent of vortices, will be visually observed with the help of stopwatch. Discharge piping from each model pump will be connected to common header and hence to suction of the slave pump used for re-circulating water in the closed circuit. Figure 1 shows experimental apparatus for the pump sump model study. It shows the model with various components like suction pipes, slave pumps, pump chamber. Figure 2 shows pillar at suction side and gate opening between the two compartments of pump unit. Figure 3 shows flow pattern during dye injection testing. Figure 4, Figure 5 shows the other details of model arrangement.

Figure 5 shows the other details of model arrangement.

Fig.1 Front View of the Sump Model Testing Rig

Fig.2 View indicating the pillar and gate Opening

Fig.3 Observation of the Flow pattern Dye Injection Test

Different combinations of pumps at rated capacity will be examined at minimum water level in the sump.The deliverable of physical model includes, flow analysis in each pump chamber, Circulations and vortices will be given in the form of swirl angle, visual effect and vortices nature, flow pattern, modifications suggested in prototype sump.

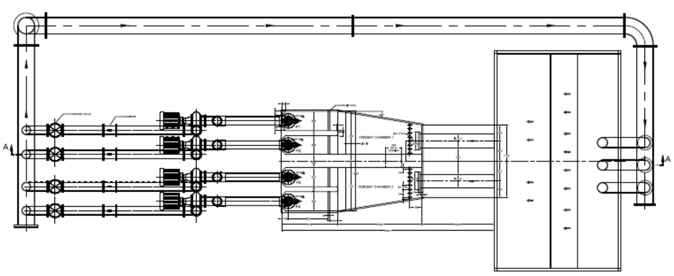


Fig. 4 Physical Sump model testing Arrangement Plan View

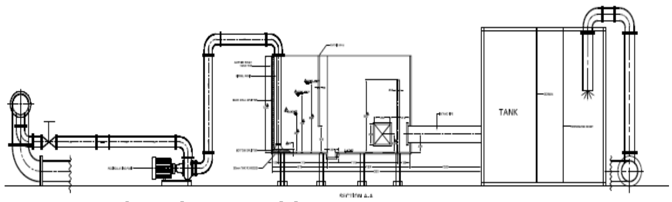


Fig. 5. Physical Sump model testing Arrangement Section View

The Reynolds number is found to be approx. 126468 and Weber number for the model is 1250. Hence kinematic similarity applies to the model.

B. *Numerical Method*

The first step of any numerical analysis is to identify the region of interest for the flow study. The geometry of the region of interest is defined. Then the mesh is created. The mesh is used for input to the physics pre-processor. The mesh is imported into the pre-processor. In this pre-processor the elements of simulation including boundary conditions are defined and fluid properties are defined. No slip shear condition is used for all the walls. The flow solver is run to produce a file of results that contains the variation of velocity, pressure and any other variables throughout the region of interest. The results can be visualized and provide an understanding of the behaviours of the fluid throughout the region of interest. The surface model of the geometry of sump is created using Pro-Engineer. The geometry is taken into ANSYS ICEM-CFD software for good quality grid generation. The grid is generated using ANSYS ICEM CFD. This grid file is further taken into ANSYS CFX-Pre for applying the suitable physics to the geometry. CFX- solver and CFX – post is used to solve and analysing the results respectively. The computational fluid dynamics results are presented in terms of vector plots and streamline plots etc. The flow quality can be visualized qualitatively by means of these results. If there is some problem in the sump, it can be clearly visualized with the help of these results. Moreover swirl angle at impeller eye location can be calculated. This gives the amount of swirl present at the entry of impeller of the pump. Thus the performance of the sump for the particular pump can be verified.

The scope of the numerical analysis was to carry out CFD analysis of the original geometry of pump sump at Minimum water level for checking of the quality of flow in the original geometry. It includes modifying the geometry to bring out optimum solution as shown in Figure 6, Figure 7 & Figure 8 as shown below.

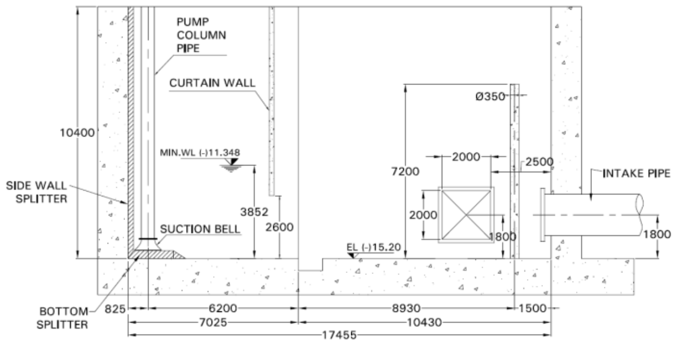


Fig.6 Pump sump intake arrangement (Section View)

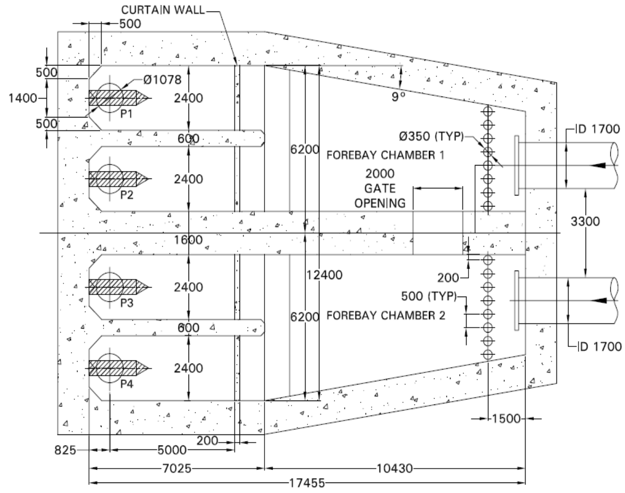


Fig.7 Pump sump intake arrangement (Plan View)

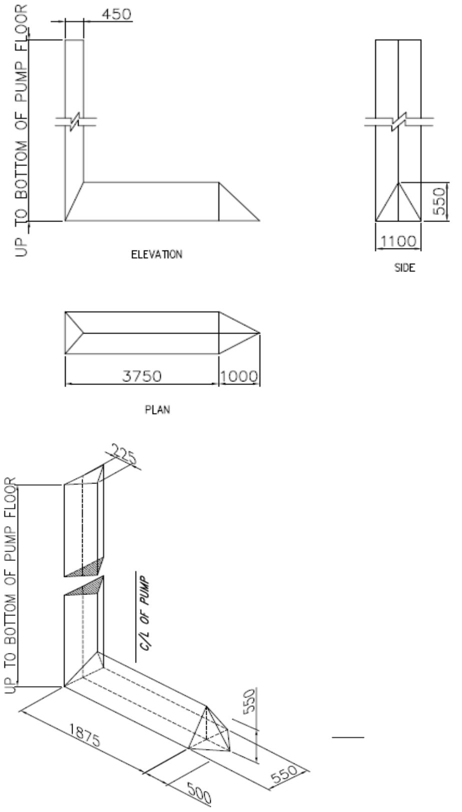


Fig.8 Different details of the triangular type vortex breaker

II. OBSERVATIONS

A. *Observations of physical model study*

Different type of pump combinations were operated at minimum water level and flow pattern studied. The maximum swirl angle was found less than 4° when all the pumps are working as per Table 1 below. The flow pattern in pump chambers was observed uniform. Intermittent Surface vortices (Type-1) were observed at upstream of curtain wall in forebay (for all pump working), but were not entering in to pump chamber. Slight Surface recirculation was observed in forebay. The same was not entering into pump chamber due to curtain wall. Overall sump performance was found satisfactory for different operation of pump combinations.

Table: 1 Pump Parameters

Parameter	Details
Discharge	7500 m^3/hr
Pump Head	22 m
No. of Pumps	3 working + 1Standby
Suction Bell Diameter	1100 mm
Inlet pipe diameter	1700 mm (2 Nos.)
Minimum water level	3.852 m

Table: 2 Observations during sump model testing

Sr. No.	Pump No.	Q Model lit/sec	Sump Water level	Swirl RPM	Remark
1	P1	12.76	Min. water level	23 cw	Slight Surface recirculation and vortices observed
	P2	12.74		41 cw	
	P3	12.68		6 ccw	
	P4	12.6		18 ccw	
2	P1	12.71	Min. water level	14 cw	Slight surface recirculation observed in forebay.
	P2	12.67		16 cw,	
	P3	12.34		8 ccw	
3	P1	12.62	Min. water level	23 cw	Surface recirculation observed in forebay
	P3	12.64		5 cw	
	P4	12.5		11 ccw	
4	P3	12.56	Min. water level	7 ccw	Slight surface recirculation observed in forebay
	P4	12.46		6 ccw	

With the help of the swirl meter revolution measured at different conditions as per the Table 2 above and model flow measured by flow meter through the pump, tangential velocity and axial velocity component were calculated respectively for the each pump. With this swirl angle is calculated for the each pump.

B. *Observations of computational studies*

The computational study carried out for the minimum water level with operating the different combinations of the pump. As per the desired flow pattern geometry is modified and different runs were taken. The presence of the vortices and recirculation is eliminated by modifying the geometry with the help of curtain wall and anti vortex type devices in the pump sump. Thus geometry is optimised. The computational fluid dynamics results are presented in terms of vector plots and streamline plots etc. If there is some problem in the sump, it can be clearly visualized with the help of these results. The flow quality can be visualized qualitatively by means of these results. The results can be visualized and provide an understanding of the behaviours of the fluid throughout pump sump.

III. RESULT AND DISCUSSIONS

During the physical sump model study, flow pattern in pump chambers was observed uniform in the modified geometry. Slight Surface recirculation was observed in forebay initially but eliminated in modified sump geometry. The same was not entering into pump chamber due to curtain wall hence not affecting the pump performance. Overall sump performance was found satisfactory for different combinations of pump operations. As per the above calculation and parameters above for the velocity component swirl angle is determined for the various conditions is noted in below table 2. While operating 3 nos. of pump swirl angle found in the order of 2°. During operation of the all four pumps swirl angle calculated is around 4°. Thus the geometry is optimised for the pump functional requirements. This is change in swirl angle is observed as common gate is opened flow transfer from one chamber to other but flow velocity reduced in the sump intake side as shown in the Table 3 below.

Table: 3 Swirl angle at 1F condition for different swirl speed (rpm)

Flow Condition	Model Flow lit/sec	swirl rpm	swirl rpm	swirl rpm	swirl rpm	swirl rpm
1.0 F	12.28	89	71	53	36	18
		5°	4°	3°	2°	1°

In computational study the comparison of results are based on the measurement of velocity at various locations, vortices, velocity profiles and the pattern of the streamlines. Computational results for the vector plots and streamline plots are as shown in the Figure 9, Figure 10 below for the 4 nos. of pump operation. Also the vector plots and streamline plots are shown in the Figure 11, Figure 12 below for the 3 nos. of pump working condition.

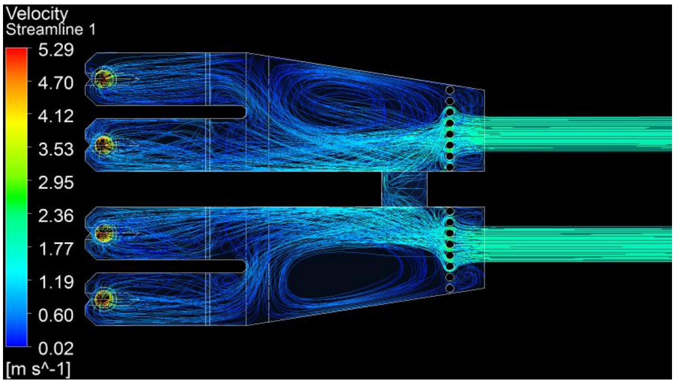


Fig.9 Plan view of CFD Analysis for Modified Geometry (4 Nos. of pump working)

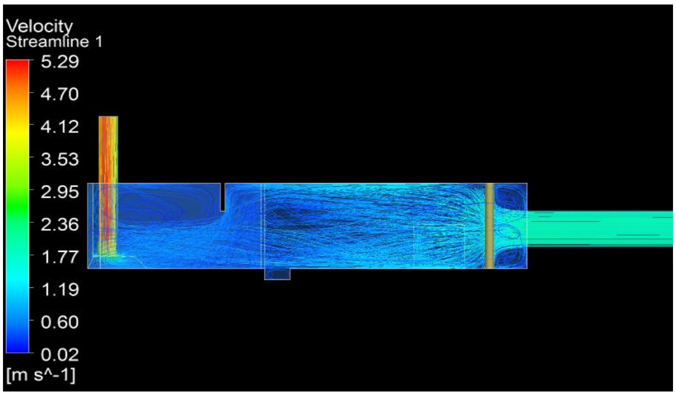
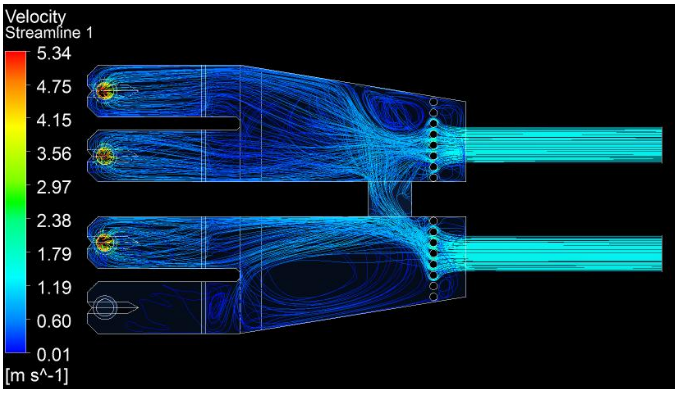


Fig.10 Section view of CFD Analysis for Modified Geometry (4 Nos. of pump working)



I.

Fig.11 Plan view of CFD Analysis for Modified Geometry (3 Nos. of pump working)

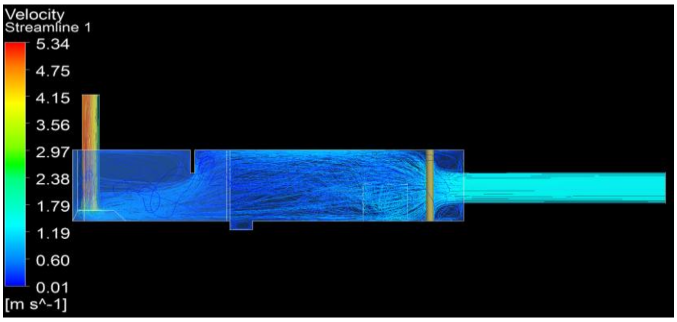


Fig.12 Section view of CFD Analysis for Modified Geometry (3 Nos. of pump working)

IV. Conclusions

In the presented study flow parameters enhancement suggested by using Fraud number and Reynolds number similarity and energy dissipating methods. Introduction of the proper curtain wall with energy dissipating devices helps in reducing the swirl and vortices in the pump inlet. Maintaining the proper clearances below the pump inlet bell, bottom and side angles, guides the flow with desired velocity. This also reduces the resistance over the surface. The submergence has important role as type 6 vortex formation occurs during minimum water level. The presented results allow us to establish some conclusions regarding the use of CFD for simulating the flow in a pump sump using the k-ε turbulence model. ANSYS Fluent is used to predict the three dimensional flow and vortices in a pump sump model. The flow pattern can be predicted in detail using CFD technique. The physical sump model is validated by comparing the experimental results with computational analysis results. The computational analysis results are found to be in good agreement with experimental results. Thus CFD model can be used as an economic and fast tool to simulate the flow field in the sump and optimising the pump sump geometry as per the pump functional requirement.

Acknowledgment

I take this opportunity to convey my deepest gratitude to my guide Prof. S.Y. Bhosale who guided me at every juncture of need. His support and cooperation helped me to finish this project work as per the requirements and guidelines.

My special thanks to Prof. S.D. Mahajan, Prof. H.N. Deshpande & all PG faculties of Mechanical Engineering Department, MCOE, Pune for their cooperation and support. I am also thankful to External Guide Mr. Pradeep Kumar Kaushik for his help and guidance during the experimental work and computational flow analysis. Experience in the field of hydraulic design helped lot for accomplishing the project work.

References

[1] V. Meena, A. Bhatt, Eldho, “Numerical simulation of pump sump flow characteristics”, Journal of Engineering and Technology, vol.2 Issue 3, pp. 49-57, Sept.2013

[2] V. Naderi, D. Farsadizadeh, A.H. Dalir, H. Arvanaghi, “Experimental study of bell mouth clearance on discharge coefficient ”, Journal of Civil Engineering and Urbanism, vol. 3 Issue 6, pp. 368-371,Oct.2013

[3] T.S. Desmukh , V.K Gahlot, “Numerical study of flow behavior in multiple intake pump sump”, International Journal of Advanced Engineering Technology (IJAET), Vol.2, Issue 2, pp. 118-121, June 2011

[4] S. A. Shabayek ,“Improving approach flow hydraulics at pump intake ”, International Journal of Civil & Environmental Engineering (IJCEE),vol.10, Issue 6, pp. 23-31, Dec.2010

[5] C.G. Kim, Y.D. Choi, J.W. 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, pp. 1-11

[6] S.N.Shukla, J.T. Kshirsagar,”Numerical prediction of air entrainment in pump intake”, Proc. 24th International Pump Users Symposium, 2008, pp.29-34.

[7] S.E.Mahgoub,” Investigating the velocity distribution in the vicinity of power plant intake intake structure”, International Journal of Applied Sciences and Engineering Research, Vol. 2, Issue 4, pp. 446-459, Aug. 2013

[8] T. C. Kueh, S.L. Beh, D. Rilling, Y. Ooi,” Numerical analysis of water vortex formation for the water vortex power plant”, International Journal of Innovation, Management and Technology, vol. 5, Issue. 2, pp. 111-115, Apr. 2014

[9] A. B. laszczyk, A. Papierski, R. Kunicki, M. Susik, “Surface vortices and pressures in suction intakes of vertical axial flow pumps”, Journal of Mechanics and Mechanical Engineering, vol. 16,Issue 1,pp.51-71, Jan. 2013

[10] P.D. Shinde, A. A. Satam, “ Cavitation effect in centrifugal pumps”, International Journal of Researchers Scientists and Developers, Vol. 2, Issue 2, pp. 20-23, Apr. 2014

