RECENT DEVELOPMENTS IN AVD'S AND PUMP INTAKE SYSTEMS – REVIEW PAPER

S. N. Gadekar[#1], Y. A. Asmar[*2], R. A. Fulmamdikar[*3], S. S. Jaiswal[*4], S. R. Patil[*5], Ajai S.[*6]

[1,2,3,4,5] *Department of Mechanical, Savitribai Phule Pune University Pune, India*
[1,2,3,4,5] *P.E.S. Modern College of Engineering, Shivajinagar, Pune- 5, Maharashtra, India*
[6] *Department of Mechanical Engineering, Anna University, Vallioor, India* [6] *Central Water and Power Research Station, Pune, India*
[1]*shivamgadekar1998@gmail.com*
[2]*iamyogesh37@gmail.com*
[3]*rajfulmamdikar03@gmail.com*
[4]*shubhangijaiswal631998@gmail.com*
[5]*sagarpatil5929@gmail.com*
[6]*ajai.mech.ceg@gmai.com*

***Abstract* - *This review paper focuses on the overview of anti-vortex devices, pump sump system, intake structure, and their performance, in recent years. The performance characteristic of pumps is developed by the manufacturer under standard conditions. But in actual practice because of a pump and tank design, irregularities at suction, and other parameters, the performance of the pump decreases. One of the major parameter which affects pump performance is vortex formation at the inlet of pump intake causing entrainment of air. These setup design parameters are unchangeable but the effect of vortex formation can be controlled with the help of anti-vortex devices. Therefore, it is essential to study and identify vortex formation in a pump sump system with and without AVD's. In this review paper, we will be focusing on vortex formation in the pump sump system and its effect on the pump system. Also in addition currently used methods to control vortex/swirl. This review could be useful for studying vortices and anti-vortex devices in the field of pump sump systems for future use*.**

***Keywords* - Vortex, Swirl, AVD, Cavitation, Bell mouth, numerical, streamlines etc.**

I. INTRODUCTION

The pump is the fluid machine that delivers energy to the fluid with the help of an impeller rotating at high speed. For efficient working of the pump, the flow properties at the pump intake must be uniform and it should not cause cavitation, flow separation, pressure loss, noise, and vibration, etc. All these effects are the result of unsteady flow and flow abnormalities at the pump intake. This abnormal flow causes the entrainment of air/air bubbles through the pump intake.

The performance of a hydraulic system like pump sump mainly depends on the geometry of its components like intake channel, forebay, and the submergence of vertical column pipe. A defective pump and sump design parameters may lead to vortices, velocity fluctuations, bulk circulations of fluid, and also induce stagnant zones during the fluid flow across its components. the formation of vortices and swirls would lead to unsteady operation of pumps and cause fall efficiency. It has been reported by experimentation that 1 % of air entrainment could lead to 10 to 15% of falls in efficiency [1].

Hence it is important to address and eliminate these unwanted flow properties to achieve the following objectives.

- To reduce the swirl formation and cavitation of the pump intake system.
- To increase the efficiency of the Pump intake system.
- To analyze the pattern of flow of water by using CFD software.
- The identifying effective shape of Anti swirl devices for pump sump system.
- Reducing vibration, surging, efficiency loss, structural damage, cavitation, flow reduction, unbalanced pump loading, noise, damage, etc.

II. LITERATURE SURVEY

Arboleda and El-Fadel [2], performed a case study of a particular pump sump model that illustrates the design features of the sump model and site constraints that affect the overall performance of pump sump systems. For the model without flow guidance, while performing experimentation they observed sub-surface vortices under all operating conditions by dye injection. From Back walls and side walls under some conditions, intermittent subsurface vortices were also observed ranging from 70 to 370.

After modifying the design of the sump, they concluded that installing flow distributor and guide structure there is a considerable amount of reduction in swirl angle which is observed around 250. This is because of the lack of uniformity in flow.

Abir ISSA Annie-Claude Bayeul-Lainé et.al**.** [3]**,** focused on the effect of pump intake location in the sump by using four conditions (varying dimensions of bell mouth, Thickness of pipe and boundary conditions.) and secondly on the effect of several inlet velocity gradients at inlet sump section with two CFD codes as STAR CCM+ and FLUENT Code. The results are presented as contours of tangential velocities, helicity, vorticity magnitudes, and streamlines. 3D Flow calculations had been performed on a sump configuration to evaluate the sensitivity of both asymmetric geometrical and velocity field configurations on the inlet intake pipe.

C. G. Kim et. al. [4], focused on vortex prediction on CFD code and verifying it experimentally. Also the causes of the occurrence of the vortices in detail. For the CFD analysis, the experimental setup model was scaled to 1:10. They used a swirl meter to identify the swirl angle by experimentation method. Several AVD's are also tested in order to control the swirl angle. The test was carried for two conditions with anti-swirl devices and without anti-swirl devices.

The results show a quite big difference between experimental and CFD results. Also, there is a reduction in the swirl angle by using Anti-swirl devices. They also concluded that vortex prediction in CFD is good and useful for designing and positioning AVD’s.

WonTae Kang et. al**.** [5], did an experimental study to analyze the effective shape of floor splitter to reduce subsurface vortices, cavitation, and swirling flow near the bell mouth in pump sump with the help of Fraud Number ranging from 0.46 to 0.36. He designed three different floor splitters having different cross-sections and dimensions viz. small and large square sections and one trapezoidal section splitter. The tank was designed on the basis of intake structure layout by HI-9.8 American National Standard.

As Per HIS Standard at Minimum water level, surface swirl and dimple observed. The results without anti-swirl devices are not as per HIS Standard as they are greater than 50. But after installing Trapezoidal AVD’s there is a reduction in swirl angle and swirls are considerably weakened. Floor splitter is good for reducing sub-surface vortices. Square shaped Floor splitter showed a large swirl angle beyond HIS Standard.

From their Experimentation, it is concluded that results obtained with the trapezoidal floor splitter are satisfactory than square-shaped AVD.

Ajai S. et. al. [6]**,** investigated on Vortex Prediction in a Pump Intake System Using Computational Fluid Dynamics under different flow conditions across the pump bay and the bell mouth section with the help of CFD code Flow 3D and verified with the experimental setup. Focusing on causes of vortex formation at bell mouth and importance of pump bay and shape of bell mouth.

The suspected areas of the vortex and its profiles were predicted by CFD and it is compared with the experiment performed on the physical model. It is also concluded that the Swirl angles predicted by CFD are Higher than Experimentation this is because of the Presence of a swirl meter in the suction pipe. In addition to that, CFD also determined free surface vortices which can be useful for designing anti swirl devices.

Won-Tae Kang, et. al. [7], suggested from their research paper that an experimental and a numerical investigation on a suction vortices behavior including cavitation in the model sump system with multi-intakes were performed. From the experiment, Type free surface vortices which occur at specified water levels were identified.

The vortices of Type A1 through Type A6 appeared in ascending order with the decreasing of water level. Minimum water levels incepting the vortices at the large flow rate were measured somewhat higher than the low flow rate. Through the multiphase flow analysis by CFD, free and subsurface vortices are reproduced and their formation, growing, shedding, and detailed swirl structures were well investigated so that it is very easy to understand the complicated swirl flow behavior. The installation of anti-swirl devices is very effective to reduce the swirl and subsurface vortices including swirl cavitation.

C. G. Kim et. al. [8], From the study on the pump sump model by the influence of AVD installation, experimental and numerical analysis have been performed and the following conclusions are achieved. According to the results from the PIV experiment, flow velocity to the flow direction shows 25% higher at the region upstream than that at the region downstream of the pump intake. The occurrence of the submerged swirl on the wall results from the different flow directions at the regions upstream and downstream of the pump intake, and the different flow velocity at the regions up to about 2times each other. According to the results from the CFD analysis, the efficiency of a mixed flow pump model shows 90.9% and 89%, respectively, when the pump sump is not included and included in the CFD analysis. By the application of the AVD in the pump sump, flow with relatively low vorticity and turbulence kinetic energy comes into the pump intake.

Johansson et. al. [9], investigated a model study on a pump sump system and computational fluid dynamics. The reason behind studying this was to identify and show, how hydraulic models are used to identify unacceptable flow conditions. In some cases unacceptable flow conditions are observed such as surface vortices, high swirl. Furthermore, results are used for the optimization of hydraulic performance in circulating water and cooling water pumps.

V. P. Rajendran et. al. [10], carried experiments on the setup of intake bay which consisted of a vertical intake pipe with a bell mouth in a rectangular channel. The experiment was taken for two conditions of water level like high submergence and low submergence along with changing the position of intake pipe from the back wall and sidewalls. They carried out detailed measurement of the velocity fields by flow visualization with the help of particle-image velocimetry (PIV) in selected sections across the free surface and wall-attached (subsurface) vortices.

Particle image velocimetry (PIV) is useful in getting the information about the nature of the vortices like the number, location, size, and strength, etc which is helpful to develop and validate the numerical model of the pump intake flows. Hence by using PIV vortices are identified and comprehensive velocity measurements were made. During the experimentation, the data in the form of images were taken for the study in which 32 images of the flow patterns were taken at the interval of 5 seconds for 16 seconds. Instantaneous and mean velocity and vorticity structures for subsurface and free surface vortices were studied at the selected regions in the tank near the intake.

After comparing these vortices for different water levels, it is found that the floor vortices are stronger among all subsurface vortices at high submergence and also there were no free surface vortices observed in this condition. This difference in strengths of vortices was due to the non-uniformity of the approach flow. In the next case of low submergence, it is found that the free surface vortices are stronger than the floor vortices in the previous case. Subsurface vortices were not measured in the lower submergence condition. Hence PIV was an effective tool for quantitative measurement of the pump intake flows and it can be replaced by a traditional dye visualization method for a more reliable design of pump intakes

Pravin Kumbhar et. al. [11], investigated vortex formation in CFD code ANSYS CFX 15.0 and Experimental validation of vortex formed in CFD code. The research was carried to study surface vortices, air entering, flow patterns. Also, the swirl angle calculation method is validated with CFD. The CFD results are represented on patterns of streamline.

In Sump Model 1 Major Problem that occurred after observing streamlines is a large rotating fluid mass on one side and dead zone on the other side. After improvements, sump model 2 showed quite good results compared to Sump Model 1. Again after some improvements in the sump, again optimized results are achieved in sump model 3. Also, in addition, the use of CFD code helped to identify location, pattern, and intensity of vortices. The authors also suggested improvements in the post-processing of CFD results for better optimization and validation of experimental results.

Takahide Nagahara et. al. [12], also experimented on a test setup to measure the velocity distribution around the submerged vortices by using particle image velocimetry (PIV) and validated the results with the velocity field obtained by CFD. The test setup consists of a rectangular channel that is divided into two sections in the upstream region of the pump sump intake and the pressure tank which is capable of generating vortices in the intake region and controlling the mean velocity and pressure at the inlet.

The measurement and calculation of the velocity profile of the swirl flow around the vortex center were carried out for different conditions. The measurement of the time-averaged velocities of the 100 instantaneous data is obtained using PIV.

The shape and location of the cavitation vortices which is validated by experimentation and the CFD tool were found to be similar in all the cases. The cavitating vortices had larger core radii and the velocity profile than the non-cavitating vortices. The core radius of the cavitating vortex is large because of the existence of the cavitating core. The circulation values of the vortices measured with the time-averaged data and the CFD has little variation in the case of the maximum velocities as the maximum velocity in the case of CFD was lower than the experimental values. But the core radii of the vortices were found to be smaller in case of the instantaneous measurement data than the time-averaged data and this is because of the unsteady movement of the vortex. Therefore, the steady-state CFD calculation cannot predict the accurate velocity profile when the vortex movement is unsteady.

Hyung-Jun Kim [13], carried out a numerical simulation study of the pump sump with anti-vortex devices (AVD) for different heights of the AVDs. In the urban lowland areas, there is an occurrence of floods due to climate changes in the rainy seasons. In this condition, the pump station is the crucial facility for flood control as it discharges the water from lowland areas to the watersheds. During pumping operation different flood conditions occur due to the changes in the river water level and the sewage flow, hence the pumping station must be able to manage the incoming flow over its design capacity.

In this study, the appropriate height of the AVD is determined along with the different flow patterns and swirls at the intake by comparing the number of different inflow conditions. The floor splitter with fillet type AVD is attached to the floor under the bell mouth in the rectangular sump model. The effective height of the AVD is determined with the help of a numerical model using the ANSYS meshing tool. The numerical model is studied for different inflow depths ranging from 0.15 – 0.30 m and AVD height is also varied according to the bell mouth diameter.

The numerical results were compared by plotting the X and Y directional velocities result for each case of the inflow depth and height of the AVD. This study of twenty test cases showed that installation of AVDs can improve the flow characteristics in most of the cases but also suggests that the flow characteristics in a sump cannot be improved in all the cases, like the installation of the AVD in the stable flow sump worsen the flow and make the flow field uncontrolled which was controlled without the AVD. Hence it is also stated that the installation of the AVD is most effective in low inflow depth conditions. The numerical result comparison showed that to improve the flow characteristics the appropriate AVD height is 0.2D.

A.R. Kabiri-Samani et. al. [14], investigated the effects of various types of anti-vortex plates on air entrainment by the experimental method. The sump is made Cylindrical shape and results are taken after allowing the system to stabilize for 15 min. It is observed that without anti-vortex plates, there is free vortex formation with an air core and audible noise occurred. Depending on the flow intensity of the swirl increased.

Keeping flow parameters the same, after installation of AVD, it is concluded that there is a reduction in the intensity of swirl also the duration of time and increase in performance of intake. They also suggested the need for more experimentation to present general design criteria for designing anti-vortex plates.

Young Joon An et. al. [15], Investigated the behavior of vortices at the suction of a centrifugal pump on CFD code CFX with different conditions and working fluid is turbine oil with a density of 868.2 kg/m3.After computation, the results show suction vortices cause more cavitation, unsteady flow also with high flow rates high cavitation was formed on the front side of blades and low cavitation on the other side. Increased vortices tend to increase the inlet flow angle.

Elzahry Farouk Elzahry et. al. [16], Studied optimum design of intake using CFD (ANSYS) for improving hydraulic performance of the pump by minimizing unacceptable flow conditions and maintaining the system below limits. The numerical experiment was carried for 4 different water levels from basin height and with 6 pumps. There is an appearance of a vortex with 5m, 4m, and 3m height water. For water level equal to basin height shows acceptable flow conditions with no vortex. Hence it is concluded that, For higher heads, the possibility of formation swirl is less.

Hyung-Jun Kim et. al. [17], Investigated by experimental method, The effect of height of Anti-vortex devices on hydraulic performance of pump sump systems, located at the urban areas where high flood crisis is to be handled. And validating it with CFD code ANSYS. It is observed that, with increase in flow velocity, vorticity also increases and thus the vortex formation in pump sump. Also AVD's with suitable height can help to maintain flow characteristics below limit.

Kadam Pratap M. et. al.[18], Studied to check suitability of pump sump for better performance of pump by physical experimentation and CFD code ANSYS FLUENT to predict 3D Flow and Vortex formation in given pump sump model. For water level below 150 mm from Minimum Draw Down Level (MDDL), Large rotating mass of fluid is observed near froebay portion and in space for future expansion maximum flow was observed also no any surface and subsurface vortices observed in both methods. For water level below 110 mm from MDDL, CFD predicted location of surface vortex near bellmouth. Author also suggested CFD ANSYS FLUENT can be used to study flow but suggested the need of improvements in post processing for more accurate and precise results.

S. Pratap et. al. [19], validated pump performance by experimental method with CFD Code ANSYS FLUENT. The main objective behind this study was to check compatibility of CFD tools for studying pump sump system and its all characteristics. Study was performed on a sump installed with 6 pumps. The results predicted by CDF calculation are almost the same even after scaling the experimental setup to 1:12. The experiment was performed on four cases by changing the number of active pumps.

In CFD, around all pumps which are running showed swirl angle less than 2.5^0. The experimental and CFD results are almost the same but for some cases because of vibration, flow fluctuations and post processing, there is a difference in results (< 1^0). Author also suggested that, This study can be further extended to test PIV, Vortex prediction, turbulence models, etc.

III] CONCLUSION

In this study, the following points are concluded

- Vortex formation is unacceptable and it can lead to various mechanical problems.
- Various CFD tools are available and can be used to calculate position, rotation, and intensity of the swirl angle. These are also useful to flow patterns, to identify dead and live zones. Although there is some error between numerical results and experimental results, this is because of vibrations, surface roughness, and methods used in pre-processing and post-processing techniques.
- Effective use of anti-swirl devices canreduce surface, subsurface swirls. Also, there is an impressive reduction in the intensity of the swirl angle with the help of the trapezoidal-shaped anti-swirl device.
- Depth of submergence have major effect on generation of swirl and it also causes variation in intensity or strength of vortices. Free surface Vortices are found stronger at low depth of submergence, whereas floor vortices are dominating in case of high depth of submergence.
- For experimentation, instead of the conventional swirl identifying method (Dye Penetration), it is effective to use Particle image velocimetry (PIV), which is a more reliable and effective method for large scale experiments.
- Although there are constraints in changing sump design, causing unacceptable flow formation. But with the help of AVD's flow parameters can be controlled.

But there is always room for improvement, in addition, it is required to find the effective shape of anti-swirl devices, Methods to identify parameters that are causing clockwise and anticlockwise rotation of vortices. Also, improvements in post-processing of CFD results are suggested. There are some issues that still need to be resolved, so we need sufficient data and guidelines regarding the designing of pump sump systems and anti-swirl devices.

ACKNOWLEDGMENTS

We thank, Prof. S. R. Patil of Modern College of Engineering, Pune for their valuable advice and discussions. We also thank, Mr. Ajai. S. of Central Water and Power Research Station, Pune for their advice and support.

REFERENCES

[1]. Ajai S., Abdul Rahiman P M, Kumar K (2016) "CFD Analysis of Multi-Intake Pump Sump System", Indian Journal of Power and River Valley Development, ISSN: 0019-5537, Issue: September-October.

[2]. Arboleda, G.; El-Fadel, M. Effects of Approach Flow Conditions on Pump Sump Design. J. Hydraul. Eng. **1996**, 122, 489–494.

[3]. Annie-Claude, Bayeul-Laine, Gérard Bois and Abir Issa. Numerical simulation of flow field in water-pump sump and inlet suction pipe, Publishing, 2010, pp.1-9. <10.1088/1755-1315/12/1/012083>.

[4]. C. G. Kim: B. H. Bang; Y. H. Lee; "Experimental and CFD analysis for prediction of vortex and swirl angle in the pump sump station model", IOP Conference Series: Materials Science and Engineering, Korea Maritime and Ocean University, 727 Taejong- ro, Youngdo-gu, Busan 606-791, Republic of Korea.

[5]. Won Tae Kang; Byeong Rog Shin, Doeg Hee Doh; "An effective shape of floor splitter for reducing sub-surface vortices in pump sump", Journal of Mechanical Science and Technology 28 (1) (2014) 175~182. Changwon National University, Changwon, 641-773, Korea, DOI 10.1007/s12206-013-1103-x , 2013.

[6]. Ajai S.; K. Kumar; P. M. Abdul Rahiman; V. S. Sohoni; V. S. Jahagirdar; "Vortex Prediction in a Pump Intake System Using Computational Fluid Dynamics", International Journal of Innovative Technology and Exploring Engineering (IJITEE) ISSN: 2278-3075, Volume-8 Issue-10, August 2019.

[7]. Won-Tae Kang and Ki Han Yu**,** An investigation of cavitation and suction vortices behavior in pump sump, Proceedings of the ASME-JSME-KSME 2011 Joint Fluids Engineering Conference AJK-Fluids2011, Hamamatsu, Shizuoka, JAPAN.

[8]. C. G. Kim; and Y. D. Choi; J. W. Choi; Y. H. Lee; "A study on the effectiveness of an anti-swirl device in the sump model by experiment and CFD", Symposium on Hydraulic Machinery and Systems, IOP Conf.Series Earth and Environmental Science 15 (2012) 072004, doi:10.1088/1755-1315/15/7/072004, 2012.

[9]. Johansson, A. E.; Stacy, P.S.; White, D.K.; Lin, F. Advancements in Hydraulic Modeling of Cooling Water Intakes in Power Plants. In Proceedings of the PWR2005 Conference, Chicago, IL, USA, 5– 7 April 2005; pp. 7–16

[10]. Rajendran V.; Patel, V; "Measurement of Vortices in Model Pump-Intake Bay by PIV",. J. Hydraul. Eng. **2000**, 126, 322–334.

[11]. Pravin Kumbhar; P. B. Patole; Haribhau Mohite; "Cfd Analysis Of Flow In Pump Sump And Physical Validation For Better Performance Of Pump", International Journal of Engineering Research and Technology 2017.

[12]. Nagahara, T.; Sato, T.; Okamura, T. "Measurement of the Flow around the Submerged Vortex Cavitation in a Pump Intake by Means of PIV", In Proceedings of the 5th International Symposium on Cavitation, Osaka, Japan, 1–5 November 2003.

[13]. Hyung-Jun Kim; Sung Won Park; and Dong Sop Rhee; "Effective Height of a Floor Splitter Anti-Vortex Device under Varying Flow Conditions", www.mdpi.com/journal/ sustainability, Sustainability 2017, 9, 285; doi:10.3390/su9020285, 16 Feb 2017.

[14]. A.R. Kabiri-Samani , S.M. Borghei, "Effects of anti-vortex plates on air entrainment by free vortex", www.sciencedirect.com, Scientia Iranica A (2013) 20 (2), 251–258 29, Oct 2012.

[15]. Young-Joon An; and Byeong Rog Shin;, "Numerical investigation of suction vortices behavior in centrifugal pump", Journal of Mechanical Science and Technology 25 (3) (2011) 767~772, www.springerlink.com/content/1738-494x DOI 10.1007/s12206-011-0108-6 4 , Jan 2011.

[16]. Elzahry Farouk Elzahry; Ashraf Ghanem, "Optimum Design of Pump Intake using CFD for Improving Hydraulic Performance", VOL 4, 2020, 57-63, eISSN 2600-8084 ARQII Publications, 14 Jan 2020

[17]. Hyung-Jun Kim; Sung Won Park; and Dong Sop Rhee; "Numerical Analysis of the Effects of Anti-Vortex Device Height on Hydraulic Performance of Pump Sump", KSCE Journal of Civil Engineering, pISSN 1226-7988, eISSN 1976-3808, www.springer.com/12205, August 30, 2016

[18]. Kadam Pratap M. and D.S. Chavan (2013), "CFD analysis of flow in pump sump to check suitability for better performance of pump", International Journal on Mechanical Engineering and Robotics, ISSN (Print): 2321-5747, Volume-1, Issue-2, pp 59-65

[19]. S. Pradeep, G. Sayantan, P.G. Prasad and M.S. Mohan Kumar (2012), "CFD simulation and experimental validation of a horizontal pump intake system", ISH Journal of Hydraulic Engineering, Vol. 18, No. 3, pp 173-185.