



ISSN 2278 – 0211 (Online)

## Assessment for Alternatives of Geometry for the Intake Side of the Sump to Enhance Performance of Centrifugal Pump

**Vibha P. Pode**

Lecturer, Dr. D. Y. Patil Institute of Engineering, Ambi, India

**Shylesha Channapattana**

Assistant Proffesor, Rajashri Shahu College of Engineering, Tathavade, India

### **Abstract:**

*"The increment in the discharge for the Centrifugal Pump tends to depreciate with the pump; especially for the higher order pumps (above 25 HP). The turbulence of vortices along with turbulent flow at the regions in the suction pipe affects the flow of water and consequently the discharge. The discharge the 'sump' is not favorably designed for aiding the intake through the suction pipe. This work would focus on Design alternatives for reducing the vortices within the suction pipe and enhancing the discharge through possible use manifold at the suction end. Alternatively, efforts would be pursued for addressing the Design of the Sump (Tank) for facilitating the flow of water at the inlet end while smoothing out the in rush of water at the extreme end of the suction pipe."*

**Keywords:** centrifugal pump, discharge, CFD

### **1. Introduction**

In any pumping system, the role of the pump is to provide sufficient pressure to overcome the operating pressure of the system to move fluid at a required flow rate. The operating pressure of the system is a function of the flow through the system and the arrangement of the system in terms of the pipe length, fittings, pipe size, the change in liquid elevation, pressure on the liquid surface, etc. To achieve a required flow through a pumping system, we need to calculate what the operating pressure of the system will be to select a suitable pump. The operating pressure of a pumped system is calculated in the SI unit of meters (m). To maintain dimensional consistency, any pressure values used within the calculations are therefore converted from kPa into m using the following conversion  
 1 kPa = 0.102 m  
 1 bar = 105 Pa

For the above system, the operating pressure or the total system head, H Total, is defined as:

$$() H \text{ Total} = HS + HD + PRT - PRES \dots (1)$$

Where,

HS = Static head (m)

HD = Dynamic head (m)

PRT = Pressure on the surface of the water in receiving tank (m)

PRES = Pressure on the surface of the water in the reservoir (m)

A centrifugal pump operated at constant speed delivers any capacity from zero to maximum depending on the head, design and suction condition. For a single family residential application, considerations other than flow and head are of relatively little economic or functional importance, since the total load is small and the equipment used is relatively standardized. For larger and more complex buildings and systems, economic and functional considerations are more critical, and performance curves must relate the hydraulic efficiency, the power required, the shaft speed, and the net positive suction head required in addition to the flow and pressure produced. The net positive suction head (NPSH) is an expression of the minimum suction conditions required to prevent cavitations in a pump. NPSH can be thought of as the head corresponding to the difference between the actual absolute pressure at the inlet to the pump impeller and the fluid vapor pressure. An incorrect determination of NPSH can lead to reduced pump capacity and efficiency, severe operating problems and cavitations damage.

## 2. Problem Statement

The pump operating at higher capacities (typically, more than 25HP) offers depreciated performance while using a normal suction pipe (combined with a normal design of the sump). The discharge of the pump is severely affected while compared with its theoretical discharge possible for that size of pump. Most of the inefficiencies are introduced due to the 'air core' and turbulence introduced due to the design of the suction side coupled with other unfavorable aspects of the Sump. The use of the electrical power, as a result, is inefficient for such purposes of lifting water in a massive volume. Consequently, the time required to deliver the output too is higher.

## 3. Aim

To improve the efficiency of the pump by considering redesign of the peripherals either to the suction or the delivery side

### 3.1. Objectives

- find the problem areas by studying the existing system
- Document the challenges to be addressed for enhancing the performance of the pump
- Consider feasibility for redesign of the suction side of the Pump
- Analyze the multi-intake manifold design using CAE software, especially in the CFD domain
- suggest the best alternative design for the intake side of the pumping system

## 4. Experimentation

The suggested design could be developed as a prototype for verification of the Mathematical or the Analytical model. This needs to be assembled with the required capacity of a suitable pump. For verification of the principle, a lower capacity pump could be engaged for experimentation. Possibility for miniaturizing the setup can be explored for reasons of development time and the implied costs for development. Alternatively, the data secured from similar experiments done in the past by the Sponsoring Company could be benchmarked for Validation.

## 5. Validation

The data captured during experimentation would be compared with the results for experimentation. A good match or concurrence would indicate a strong basis for validation of the thesis work. Appropriate correction factors needs to be considered while making the comparison due to effects of the environment or the variation in the physical setup or the conditions of experiment.

## 6. Literature Review

The Jong-Woong Choi et.al tested the sump model to examine the flow structure around pump intake. In this study, flow uniformity according to the flow distribution in the pump intake channel is examined to find out the cause of vortex occurrence in detail. A multi-intake pump sump model with 7 pump intakes and a single-intake pump sump model are adopted for the investigation. Furthermore, effectiveness of antishubmerged vortex device (AVD) for the suppression of the vortex occurrence in a single pump intake, as well as in a multi-intake pump sump model has been examined by the methods of experiment and numerical analysis. The results show that most high value of flow uniformity is found at the inlet of pump intakes 3 and 5 in the multi-intake pump sump with 7 pump intakes. Strong submerged vortex can be successfully suppressed by AVD installation on the bottom of pump intake channel just below the bell mouth.

Tanweer S. Desmukh & V.K Gahlotuses studied the commercial CFD package ANSYS CFX-10 to predict the three dimensional flow and vortices in a pump sump model. The CFD model predicts the flow pattern in detail and the location, and nature of the vortices. However, considerable post-processing of the basic data is needed to fully comprehend the details of the flow. Thus CFD model can be used to study the effect of various parameters and hence can become an important tool for optimization of pump sump geometry.

Kadam Pratap M. and D. S. Chavan studied the flow characteristic in a pump sump of physical model by using Computational Fluid Dynamics (CFD) code FLUENT. The experimental procedures include the data collection using a flow meter and 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 are carried out in FLUENT , and flow is analyzed with the help of velocity stream lines and vector plot and velocity contour at the entrance of pump chamber, in CFD-POST software and concluded on experimental and cfd results.

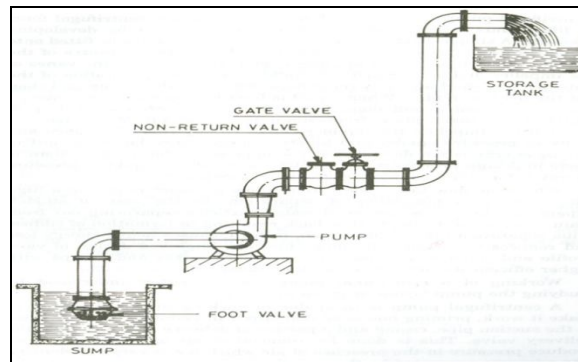
A C Bayeul-Lainé, G Bois and A Issare produced the flow pattern and confirm the geometrical parameter influences of the flow behavior in such a pump. The numerical model used solves the Reynolds averaged Navier-Stokes (RANS) equations and VOF multiphase model for two cases. In the validation of this numerical model, emphasis was placed on the prediction of the number, location, size and strength of the various types of vortices. Previous studies, without simulation of air entrainment, have shown the influence on a single type of mesh with different cell numbers, different intake pipe depths and different water levels, for two turbulence models closure.

Shazy A. Shabayek conducted an experiment on a physical hydraulic model of a circulating water pump sump structure. The objective of the hydraulic model study was to evaluate the performance of the initial design of the pump sump and to develop modifications to eliminate flow problems such as severe vortexing, intense swirl, or uneven flow distribution at the pump bell. Proposed modifications to the sump included the installation of sidewall fillets, back-wall fillets and center floor splitters. With these modifications installed in

the model, flow pre-swirl, vortex activity and pump throat velocities were all within acceptable limits for the range of operating conditions examined in the model.

Samir A. S. Ibrahim tested the design of the pump intake and to investigate any modifications on the design which may be needed to enhance the efficiency of the pump intake structure. Also some sensitivity tests were done at several water levels of the pump house. The model was operated at high, minimum water conditions and at different operation modes. Physical model is expected to help in evaluating the operation, to aid in design or modifying of the intake structure and to test any recommended modifications, if needed. The results of the model test is used to determine if the initial design of the pump sump structure would be susceptible to surface and sub-surface vortices, preswirl entering pumps and excessive fluctuation in the velocities at the pump bell throat according to Hydraulic Institute (HI) specifications. The results of the tests showed that the rotations of the roto-meters are within the allowable limits specified by the HI criteria in all operation modes. Cecilia Lucino, Sergio Liscia Y Gonzalo Duró verified the ability of a commercial computational fluid dynamic (CFD) code to predict the formation of vortices in a pump sump. It was intended to identify vortices of diverse origin and intensity in a geometrically simple pump sump of which experimental results under the same operating conditions are known. Calculated velocities correlate well to trends and magnitudes of measured ones, whereas the maximum values of vorticity calculated are several orders of magnitude higher than those measured, which is explained by the characteristics of measurement in the physical model. The representation of total vorticity, in terms of its absolute value, and selected as an equipotential surface, may turn into a very useful tool to visualize and follow the actual location, trajectory and time variation of concentrated vortices.

## 7. General Pump



## 8. Design Criteria

When flow is zero pump head will be highest with incessantly decreasing head as flow increases to insure steady operation of one or multiple pump in parallel, at all loads.

Pump will operate continuously without overheating or intolerable noises at minimum recirculation flow and at all loads without internal flashing.

Required for prior condition will be made in pump design for expansion of

- Rotor and casing are related to each another.
- Casing related to the base of the system.
- Pump rotor and shaft of the driver are related to one other.
- Outer and inner casing for double casing pumps.

The parts which rotate will be balanced statistically and dynamically for all speeds.

At all outputs axial as well as radial balance of the rotor will be provided by pump design.

One end of the pump shaft will be nearby for portable tachometer measurements.

Each pump will be provided with a pump reheat system so that when it is used as a supply it can be hot, ready for quick startup.

This is done by connecting orifice from the common discharge header to the pump discharge inside of the stop and check valve.

Then hot water can flow back through the pump and open inlet valve to the common suction header, thus keeping the pump at operating temperature.

Pump will be designed in that manner it will start safely from a cold start to full load in 60 seconds in an emergency, although it will normally be warmed before starting as described above.

## 9. Conclusion

The data captured during experimentation would be compared with the result for experimentation. A good match or concurrence would indicate a strong basis for validation of the work. Appropriate correction factors needs to be considered while making the comparison due to effects of the environment or the variation in the physical setup or the condition of experiment.

**10. References**

1. "Investigation of Hydraulic Characteristics of a Volutetype Discharge Passage based on CFD" Honggeng Zhu, Rentian Zhanga, Guoqiang Luo, Bin Zhang, Yangzhou University, 131 M Jiangyang RD, Yangzhou, 225009,
2. P. R. ChinabJiangsu Surveying & Design Institute of Water Resources Co., Ltd., 131 M Jiangyang RD, Yangzhou, 225009, P. R. ChinacTaihu Region Water Project Administration Department, 24 East Taihu RD, Suzhou, 215128, P. R. China
3. KadamPratap M. and Chavan, "CFD Analysis of Flow in Pump Sump to Check Suitability for Better Performance of Pump", Volume-1, Issue-2, 2013.
4. Bin Cheng, Yonghai Yu, "CFD Simulation and Optimization for Lateral Diversion and Intake Pumping Stations", International Conference on Modern Hydraulic Engineering, 2012.
5. Samir A. S. Ibrahim, "Evaluation of the Flow Characteristics in the Intake Structure and Pump Sumps Using Physical Model", Nile Basin Water Science & Engineering Journal, Vol.4, Issue 1, 2011.
6. Jong-Woong Choi, Young-Do Choi , Chang-Goo Kim and Young-Ho Lee, "Flow uniformity in a multi-intake pump sump model", Journal of Mechanical Science and Technology, 2010.
7. Shazy A. Shabayek, "improving approach flow hydrodynamics pump intake", International Journal of Civil & Environmental Engineering IJCEE-IJENS Vol:10, 2010.
8. A C Bayeul-Lainé, G Bois1 and Aissa, "numerical simulation of flow field in water-pump sump and inlet suction pipe", earth and environmental science 12, 2010.
9. Tanweer s. Desmukh&v.kgahlot, "simulation of flow through a pump sump and its validation", ijrras4 (1),
10. july 2010.
11. Cecilia Lucino, Sergioliscia Y Gonzaloduró, "vortex detection in pump sumps by means of CFD",