International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 7, Special Issue of ICRTET-2023, pp 202-205, June-2023

International Conference of Recent Trends in Engineering & Technology (ICRTET-2023)

Experimental Study and Simulation of Test Loop Flow Characteristics for Vertical Pumps

[1]Prof. Rahul Kulkarni, [2]Mr. Vijay Machhindra Surwase

[1]Professor, Mechanical Engineering Dept. Siddhant College of Engineering, Sudumbare, Pune-412109, India
[2]Student, Mechanical Engineering Dept. Siddhant College of Engineering, Sudumbare, Pune-412109, India
Authors E-mail: [1]rahulrkul@gmail.com, [2]vijaymechengg@gmail.com

***Abstract* - The efficiency and performance of pump involving multiple pumping units depends not only on the efficiency of the pumping units but also on the proper design of the test loop. The proper design of pump intake 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 design and optimization of sump geometry can be reduced to a large extent through CFD studies. However, writing a separate code for each new product is not feasible. Hence this work is aimed at determining the feasibility of commercial CFD software as a design optimization tool for pump sumps. In the present study commercially available software ANSYS CFX has been used for CFD analysis of flow conditions in a pump sump and the results obtained are found to be in good agreement with the experimentally observed flow patterns.**

Keywords: Pump Sump, Air entering, SwirlAngle, computational fluid dynamics (CFX 15.0).

I. Introduction

The main aim of sump is to provide water with Swirl Free, air entering, uniform velocity during the pump operation, abnormal flow phenomena such as cavitation, flow separation, pressure loss, vibration and noise occur often by flow unsteadiness and instability. It is an accepted fact that faulty design of pump sump or intake is one of the major causes of unsatisfactory operation of pumps in any pumping plant. The adverse flow conditions at a pump intake lead to occurrence of air entering, swirl and vortices, which in turn reduce the pump efficiency, induce vibrations and excessive bearing loads and lead to other operating difficulties. Thus at present model studies are the only tool for developing a satisfactory design of a pump sump, additional modification such as vortex suppression devices (Cruciform), flow straightner, change the position of curtain wall. According to the HI standard or ASME criteria for a pump sump design. The objective of the present work is to close this gap by evolving a method to quantify the swirl angle, Air entering and uniform velocity

Design criteria

The efficiency and performance of pumping stations involving multiple pumping units depends not only on the efficiency of the pumping units but also on the proper design of the pump sump. The proper design of pump sump is not an easy task because of the various site-specific geometrical and hydraulic constraints. Hydraulic Pump sumps are designed to provide Air entering, surface vortices, swirl free flow to the pump. The degree of swirl is measured in physical model tests using a swirl meter and a quantity known as swirl angle is generally measured. Remove air entering when change the position of curtain wall. The present paper presents a novel method to compute the bulk swirl angle using the local velocity field obtained from computational fluid dynamics (ANSYS CFX 15.0) data. The basis for the present method is the conservation of angular momentum conservation. By carrying out both numerical and experimental studies of air entering, surface vortices,flowpattern,swirl angle calculation method is validated Further the effect of vortex suppression devices (Cruciform) in reducing the swirl angle, air entering is also demonstrated.

Geometry of Computational Model

Realistic and Suitable Boundary conditions will be applied to the ready model in the CFX. The boundary conditions depend upon 1. Single pump capacity 2. Number of working pumps.

The domain was specified as a Stationary Fluid domain with working fluid as normal water and reference pressure as 1atm. The turbulence model was selected as SST model. As boundary conditions for the steady state calculation, and isothermal process is considered. Total flow rate and null pressure-inlet condition at the inlet and at the outlet of pipe each pump operating flow rate is applied. considering the equal mass flow condition for similar pump in the suction line. Water level is kept at Defined level. As the wall conditions, free-slip condition is applied to the upper side of pump sump

International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 7, Special Issue of ICRTET-2023, pp 202-205, June-2023

International Conference of Recent Trends in Engineering & Technology (ICRTET-2023)

model passage. No-slip condition is applied to side and bottom walls and at pipe outside and inside.

II. CFD STUDIES

From a purely numerical perspective the geometric complexity of the problem is such that it demands the full power of modern computational fluid dynamics (CFD) to solve the equations of motion and turbulence models in domains that involve multiple surfaces. Additional difficulties are associated with modeling free surface and vortex phenomenon, the physics of which is not yet fully understood. Inspite of the practical importance of the problem, the literature on numerical modeling of pump intake flows is rather limited. Review of the available literature reveals that site specific computational studies have been undertaken for certain projects [4,9]. The only generalized study has been conducted at IIHR [1,2] and that too pertains to development of CFD code for simple rectangular sump having one and two pumps. The developed software cannot be used as general software for CFD analysis of any intake structure. In the present work an attempt has been made to simulate and predict the flow conditions such as vortices and swirl for multiple pump intakes in a single sump, with an aim to determine the viability of, commercially available computational fluid dynamic software – ANSYS CFX as an important design optimization tool for intake sumps.

III. GEOMETRY OF COMPUTATIONAL MODEL

Computational study was conducted for the pumping system of a cooling tower having three pumps, of which the two end pumps were working while the central pump was a non-working standby pump. The layout includes a leading channel, approach channel, forebay, pump sump and intake. A model of the prototype at a scale of 1:11was used for hydraulic analysis. The geometry of the simulation model starts with the inlet to the sump followed by a short approach section and a vertically sloping section which ends in an expanding forebay. After the forebay is the rectangular portion of the sump consisting of three identical pump bays separated by piers. Towards the end of each of the bay is placed the suction pipe of a pump at required clearances from the boundaries. The length of the suction pipes is extended above the sump boundary to some distance. Figure 1 gives the schematic diagram of the model in plan and elevation showing all the basic dimensions.

The computational investigations were performed using ANSYS ICEM CFD 10.0 and ANSYS CFX-10.0 softwares. ANSYS ICEM CFD 10.0 was used for modelling and mesh generation while the analysis was done using ANSYS CFX10.0. The inputs and outputs of both the softwares are in easily accessible formats enabling full integration with any CFD software. For the CFD model in the present study, volumetric meshing with unstructured tetra meshing option was adopted for grid generation in the pump sump geometry. In general the mesh generated for different variants had about 8 to 12 lakh elements with the number of nodes varying from 1 to 2 lakhs. The physics of the simulation domain was defined in CFX-Pre, the preprocessing module of ANSYS CFX. The domain was specified as a Non-Buoyant, Stationary Fluid domain with working fluid as water at 25°C and reference pressure as 1atm. The turbulence model was selected as K-Є model. The inlet section at the entrance to the sump was specified as inlet boundary with mass flow rate as 137kg/s and flow direction as normal to the boundary. The outlets of the three pipes were specified as outlet boundaries with relative average static pressure as 0atm. The flow regime at both inlet and outlet boundaries was specified as subsonic. All the outer walls of the flow region and the internal walls (pump columns below free surface) were specified as the boundary type wall with flow condition as - no slip. The free surface of fluid was specified as a symmetry boundary i.e. a stress free plane of symmetry or surface across which no flow takes place.

The ANSYS CFX-Solver module of ANSYS CFX-10 was used to obtain the solution of the CFD problem. The solver control parameters were specified in the form of solution scheme and convergence criteria.

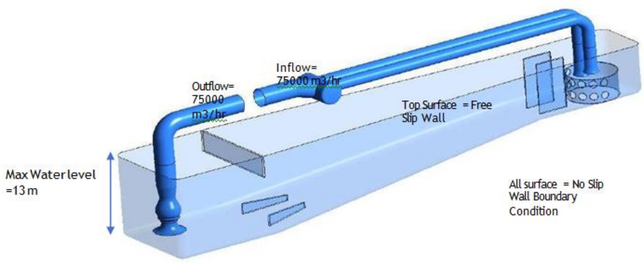

Figure 1. Modelled basic sump geometry

Upwind scheme was specified for the solution while for convergence the residual target for RMS values was specified as 10^{-6}. With the above specified convergence criteria it took about 48hrs for the solution of one simulation variant. The results of the CFD analysis were analysed using CFX-post, the post processor module of ANSYS CFX-10. Initially, numerical simulation of flow through the sump model (scale 1:11) was performed and the results analysed. Results of numerical simulation of flow in the original model showed flow disturbances in the forebay as well as the central pump bay of the sump which in turn create many other undesirable flow conditions. To minimize these disturbances modifications

International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 7, Special Issue of ICRTET-2023, pp 202-205, June-2023

International Conference of Recent Trends in Engineering & Technology (ICRTET-2023)

were done in the sump geometry. The configuration of the model showing acceptable flow conditions was then replicated in the laboratory for experimental validation

IV. ANALYSIS OF SIMULATION RESULTS

The results of the computational simulation can be analyzed using number of variables. In this study it has been restricted to the comparison of results based on the pattern of streamlines of flow and the velocity profiles. The major problem revealed through the study of the streamlines of flow is the formation of a large rotating fluid mass, in the central bay with the non working pump.

The streamline pattern in the vertical plane parallel to the sump axis (Figure 2) shows that a very large rotating mass of fluid is created in the rectangular portion along the centerline of the sump.

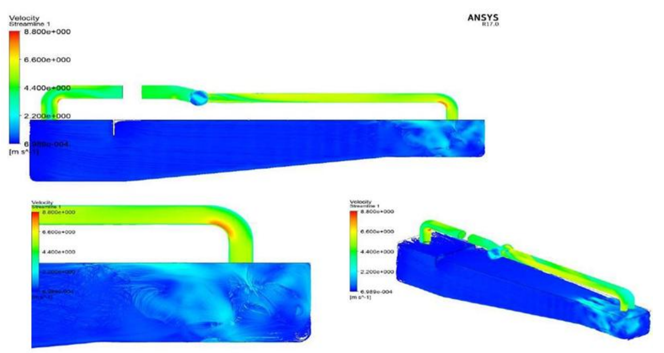

Figure 2. Streamline pattern in vertical longitudinal planes in Original Model

This rotating mass intrudes about 0.8m into the expanding forebay of the sump and on the downstream side it just intrudes into the pump bays. The extent of the rotating mass is maximum along the centerline and it reduces gradually as we move towards the boundaries. This is due to the presence of a non working standby pump in the central bay. When the high velocity flow near the channel bottom encounters the obstruction due to pump column it is forced to turn back and hence a rotating mass of fluid is created. In the vertical direction the rotating mass extends from the free surface to about 0.2m above the sump bottom. The magnitude of velocity is minimum (about 0.05m/s) at the center of the mass and increases towards the periphery reaching a maximum of about 0.1m/s. As the high velocity flow beneath the rotating mass reaches its downstream end, the flow moves towards the free surface (in the sudden empty space) and then turns towards the backwall to form another rotating mass in the central bay. In the side bays, as the extent of the rotating mass is less the distance of upward movement as well as the velocity of flow moving upward is less and the second vortex is not formed completely.

The streamline pattern in the sump shows that the upper half of the vortex flow opposes the incoming flow. Hence when the directly coming flow from upstream meets the rotating mass, streamlines in upper half of the rotating mass are forced to turn back and enter the side bays. These returning flows in the sump compress the streamlines coming from upstream towards the boundary. This is evident in the streamline pattern in the horizontal plane at various depths (Figure 3). The effect of returning flow maximum at the free surface and minimum at the center of the vortex. In lower half of the rotating mass the direction of flow is same as that of the flow coming from upstream. Therefore there is no turning back of flow and the entire flow moves downstream towards the pump columns.

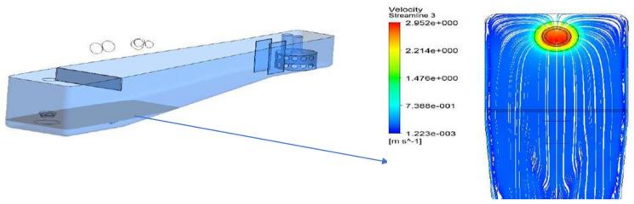

Figure 3. Streamline pattern in horizontal planes in Original Model

V. EXPERIMENTAL VALIDATION

A physical model confirming to the configuration (Figure 4) of the variant showing acceptable results was reproduced in the laboratory. This model was tested with the aim of validating the computational results experimentally. The experimental setup was fabricated as a recirculating system with water from the pump bays in the sump being pumped by vertical turbine pump to the stilling tank. Water from the delivery pipes of the two pumps is released in the corners of the far end of the stilling tank From there it flows back to the pump bays through the approach channel and forebay of the sump. For reducing the turbulence and straightening the flow before it enters the sump, two perforated baffles and a screen have been provided in the stilling tank. The sump takes off centrally from the stilling tank such that its top level matches the top level of the tank. The base of sump is supported through out, on staging made of crossed iron bars in angle iron frame. The stilling tank, approach channel, forebay, pump bays and piers have been fabricated from welded MS sheets. For discharge measurement, sharp edged orifice meters with d/D ratio 0.73 have been provided in the delivery pipe of each pump, with sufficient straight length of pipe both on the upstream and downstream side. The orifice meters were calibrated before conducting the tests. Glass windows were provided in the sidewalls and backwall (one in each pump bay) of the sump to facilitate visual observations. Taking photographs of flow patterns on the surface and inside water is not possible with the normal

International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 7, Special Issue of ICRTET-2023, pp 202-205, June-2023

International Conference of Recent Trends in Engineering & Technology (ICRTET-2023)

flash gun of a camera as the light from the flash gets reflected from the water surface. Hence arrangements were made to light up the water between the piers and guide piers from the windows in the sidewalls by means of Halogen bulbs.

Figure 4. Geometry of the model used for experimental Investigations

VI. CONCLUSIONS

The commercial CFD package ANSYS CFX-10 was used 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.

REFERENCES

[1] Constantinescu G.S., and Patel, V.C. (1998), “Numerical model for simulation of pump-intake flow and vortices”, ASCE Journal of Hydraulic Engineering, Vol.124, No.2, 123-124.

[2] Constantinescu1, G.S. and Patel, V.C. (2000), “Role of Turbulence Model In Prediction of Pump-Bay Vortices”, ASCE Journal Of Hydraulic Engineering, Vol.126, No.5, 387-390.

[3] Hydraulic Institute standards 1975. Centrifugal, Rotary and Reciprocating Pumps. 13th edition, Cleveland, Ohio.

[4] Joshi, S.G. and Shukla, S.N. (2000), “Experimental and Computational Investigation of Flow through a Sump”, Pumps & Systems Asia 2000. Nakato Tatasuaki., Weinberger Marc. and Logden Fred. (1994), “A Hydraulic model study of Korea Electric Power Corporation Ulchin Nuclear Units 3 and 4 circulating-water and essential-service-water Intake structure”, IIHR Technical Report No 370.

[5] Nakato,T. and Darian De Jong (1999), “Hydraulic Model study of water-Intake structures for Meizhou Wan Power station, The peoples Republic of China”, IIHR Technical Report No 402.

[6] Nakato,T., Darian De Jong and Brosow,Volker (1999), “Hydraulic Model study of Red Hills generating facility circulating water pumps”, IIHR Technical Report No 408.

[7] Prosser, M.J 1977. The Hydraulic Design of Pump Sump and Intakes. British Hydromechanics Research Association.

[8] Shukla,S.N. and Kshirsagar,J.T. (1999), “Sump Model simulation using CFD tools”, International CFX Users Conference, German.

Citation of this Article:

Prof. Rahul Kulkarni, Mr. Vijay Machhindra Surwase, “Experimental Study and Simulation of Test Loop Flow Characteristics for Vertical Pumps” in proceeding of International Conference of Recent Trends in Engineering & Technology ICRTET - 2023, Organized by SCOE, Sudumbare, Pune, India, Published in IRJIET, Volume 7, Special issue of ICRTET-2023, pp 202-205, June 2023.

****