SIMULATION OF FLOW THROUGH A PUMP SUMP AND ITS VALIDATION

Tanweer S. Desmukh & V.K Gahlot

Civil Engineering Department, M.A.N.I.T,Bhopal
E-mail: tanweer_sultanah@yahoo.com, gahlot_vk@rediffmail.com

ABSTRACT

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 Intake sump. 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: *Sump, numerical simulation, streamline pattern, swirl*

1. INTRODUCTION

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 swirl and vortices, which in turn reduce the pump efficiency, induce vibrations and excessive bearing loads and lead to other operating difficulties.

The flow conditions at entry to a pump depend upon flow conditions in approach channel, sump geometry, location of pump intake with respect to the walls, velocity changes and obstructions such as piers, screens etc., and rotational tendencies in flow produced upstream of the pump bays. Analytical determination of the flow conditions in a sump is not an easy task due to the complex nature of the flow. Moreover the analytical solution may not completely predict the actual conditions in the sump due to the assumptions made for simplifying the analysis. Thus at present model studies are the only tool for developing a satisfactory design of a pump sump, yet numerical simulation is a very good facility for reducing the time and cost involved .in the design process.

2. DESIGN CRITERIA

Traditionally sump design has relied upon Hydraulic Institute pump standards [3] for obtaining the sump dimensions and pump position relative to the sump walls. These design guides originated and are extrapolated from experience with smaller pumps where approach flow conditions especially subsurface vorticing are not as critical as they are for large capacity pumps employed today. A more comprehensive guide to pump design given by Prosser [8] is based upon research performed at BHRA. This guide gives sump dimensions and relative position of the pump in terms of the dimensionless ratios of the distance in question to the pump bell diameter.

Application of BHRA guidelines or Hydraulic Institute standards to design a major sump does not generate a problem free sump but provides only a basis for the initial design. As there are no specific guidelines or criteria for design of trouble free intakes, the most common solution to potential problems in new deigns and rectification of problems observed in existing designs is to construct a scaled model in a laboratory, observe and investigate the flow therein and propose modifications to the intake geometry. Further additional devices in the form of floor splitters or cones, backwall splitters, fillets, surface beams, guide vanes etc, aimed at controlling the vortex and swirl formation may be required to achieve a design which meets the performance criteria. Considerable prior experience and ingenuity are required to solve the problems by this method and the cost and time required are also significant.

As the experiments are expensive most of the studies in this field pertain to solutions of site specific problems. IIHR has conducted many such studies [5,6,7] for design of new sumps as well as for improving the design of existing sumps. However the results do not lead to any general criteria for design but contribute only indirectly to the development of a general design methodology. In view of this, researchers are now venturing into the field of numerical modeling of pump intake flows using Computational Fluid Dynamics.

3. 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.

4. 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.

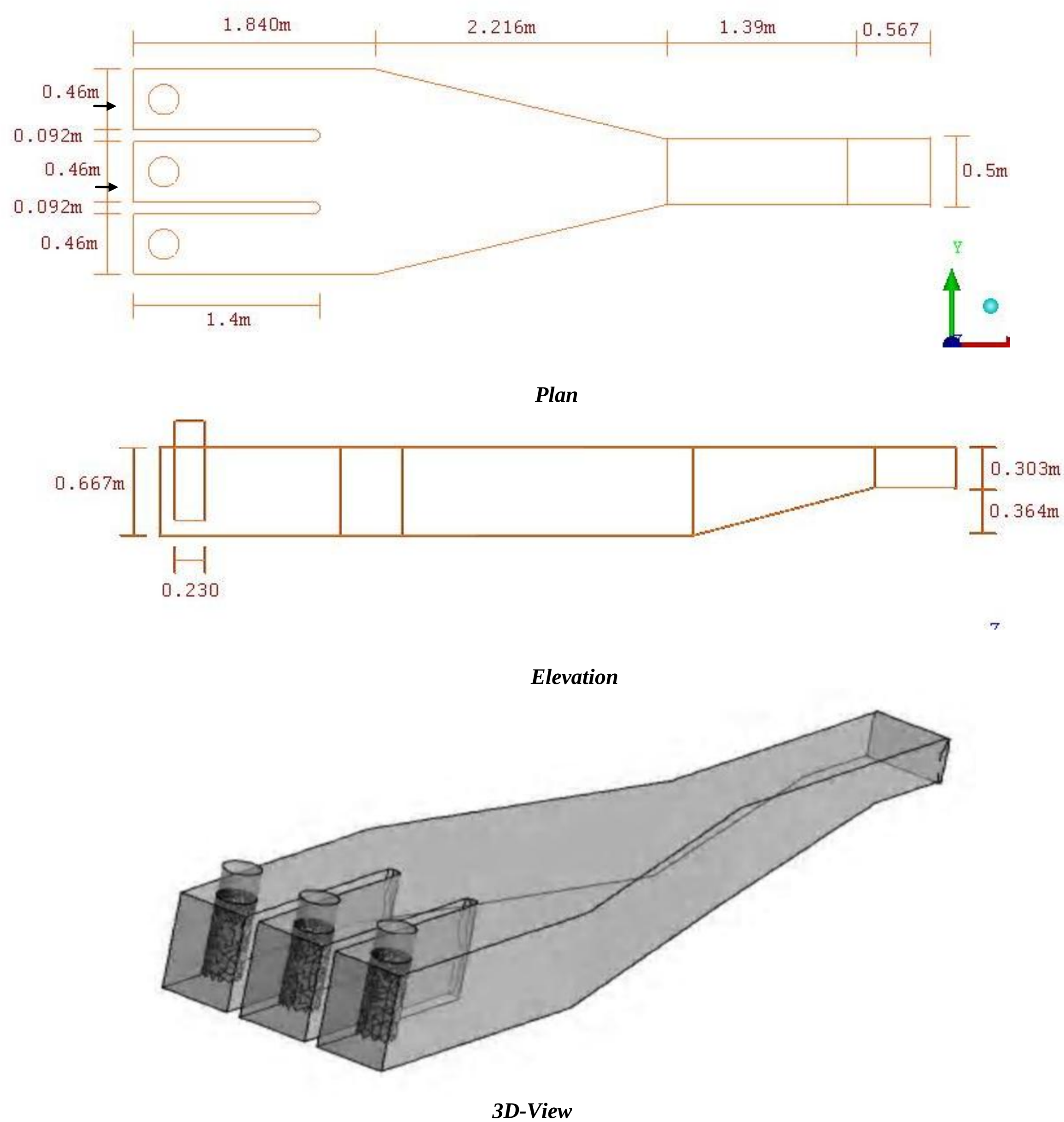


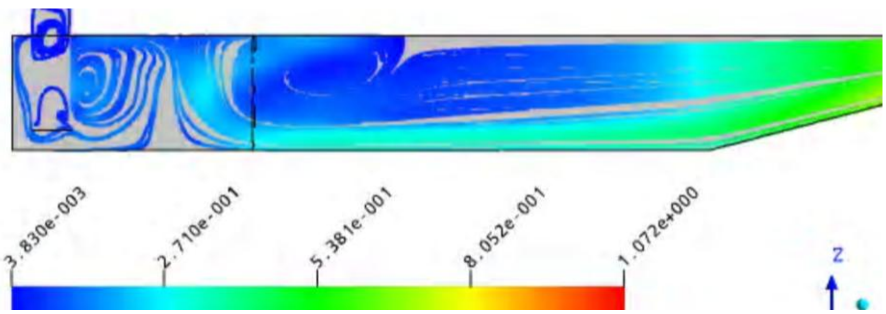
Figure 1. ***Modelled basic sump geometry***

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.

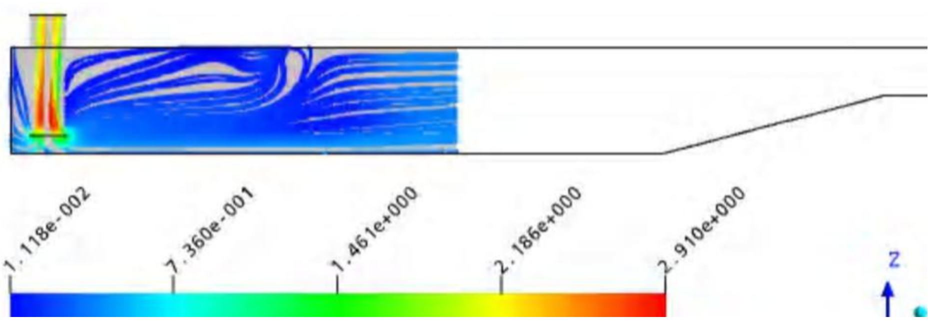
The results of the CFD analysis were analysed using CFX-post, the post processor module of ANSYS CFX-10. Initially, numerical simulation of flow through the sump model (scale 1:11) was performed and the results analysed. Results of numerical simulation of flow in the original model showed flow disturbances 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 were done in the sump geometry. The configuration of the model showing acceptable flow conditions was then replicated in the laboratory for experimental validation

5. 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.



Along centerline of sump

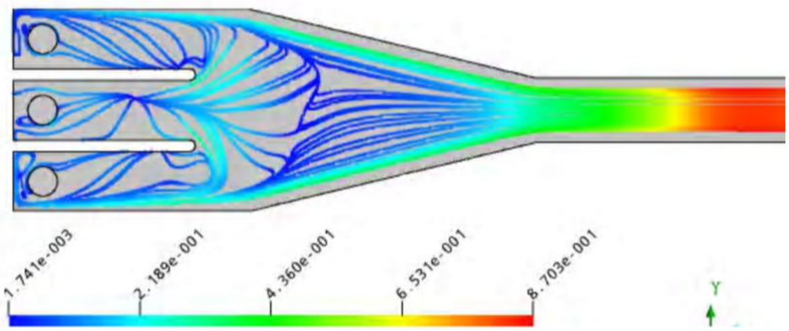


Along pump axis in end bay

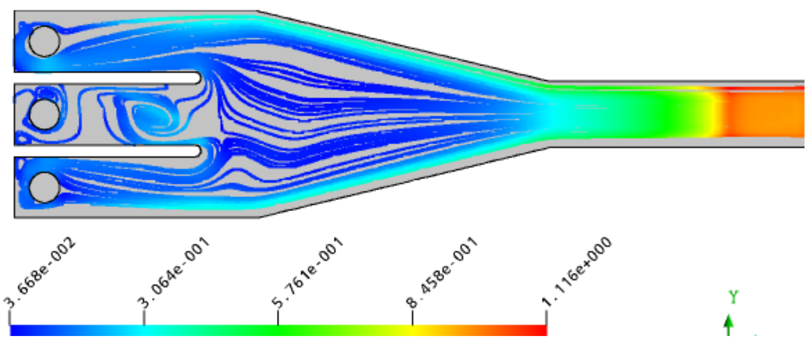
Figure 2. ***Streamline pattern in vertical longitudinal planes in Original Model***

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 flows is 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.



Horizontal Plane at free surface



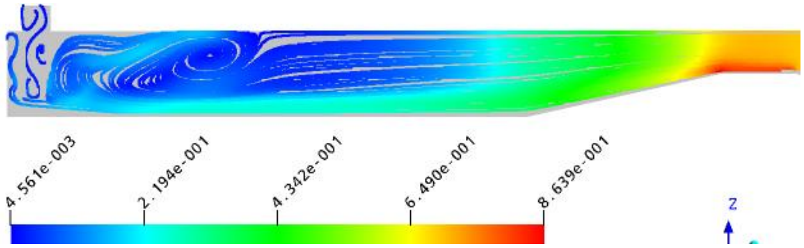
Horizontal Plane at 0.4m from bottom of sump

Figure 3. ***Streamline pattern in horizontal planes in Original Model***

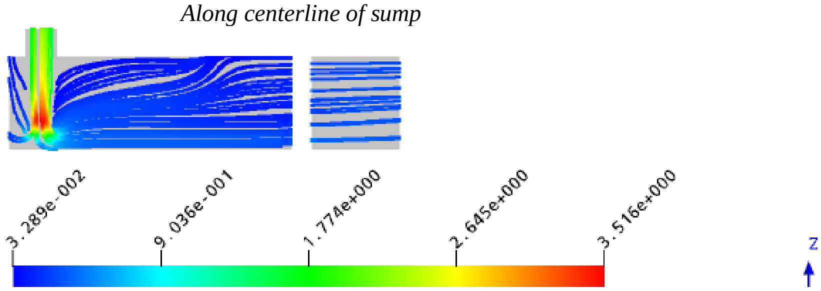
The velocity vector plot in the vertical pipe column (Figure 6) shows the presence of angular velocity and swirl in the flow. This results in irregular velocity variation (1.2m/s - 2m/s) at the inlet. The plot also shows an increase in the magnitude, just beyond the inlet which is due to the reduction of flow area in the bell mouth.

The above observations show the presence of undesirable flow conditions in the sump. Improvement in the flow conditions can be achieved through suitable modifications in sump geometry and/or providing appurtenances. In the present case, to minimize these disturbances, a number of variants of the original sump model with modifications in different elements of the sump geometry such as angle of horizontal expansion, vertical slope, pier length etc were tried out and the numerical simulation for each of these variants was performed. Inspite of the various modifications in the sump geometry and provision of additional devices the rotating mass in the central bay could not be eliminated completely. However each modification was aimed at reducing the extent of the rotating mass in the central bay and thus making the flow conditions in the end bays more uniform.

The results of the finally selected configuration showed improved flow conditions from amongst all the variants. For the final model the streamline pattern in the vertical plane along the longitudinal axis of the sump shows that instead of two rotating masses there is only one mass now. Also, its intrusion in the forebay is reduced from 0.8m to 0.15m (Figure 4).The streamline pattern in the horizontal planes (Figure 5) show that there is a reduction in the returning flows entering the end bays thus reducing the compression of streamlines towards the sidewalls. This results in an increase in the flow entering the pump bays directly and a corresponding decrease in the flow returning from the central bay which in turn makes the flow more uniform. There is an overall increase in the flow entering the end bays especially in the lower half of the depth which in turn results in an increase in the magnitude of velocities near the bottom of the channel in the end bays

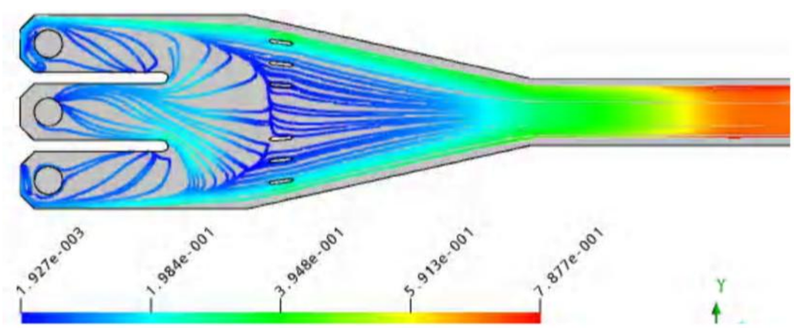


Along centerline of sump

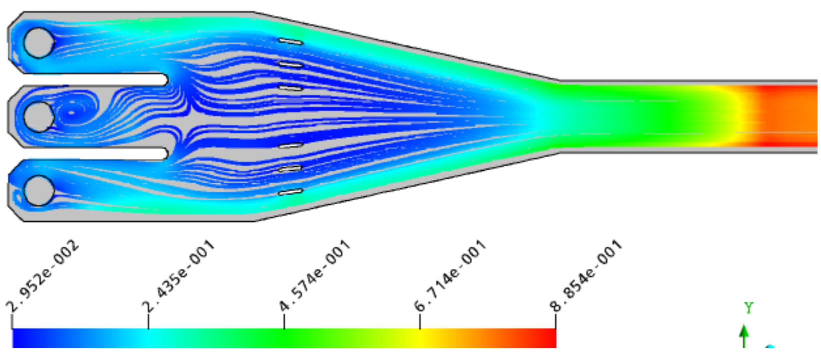


Along pump axis in end bay

Figure 4. ***Streamline pattern in vertical longitudinal planes in Final Model***

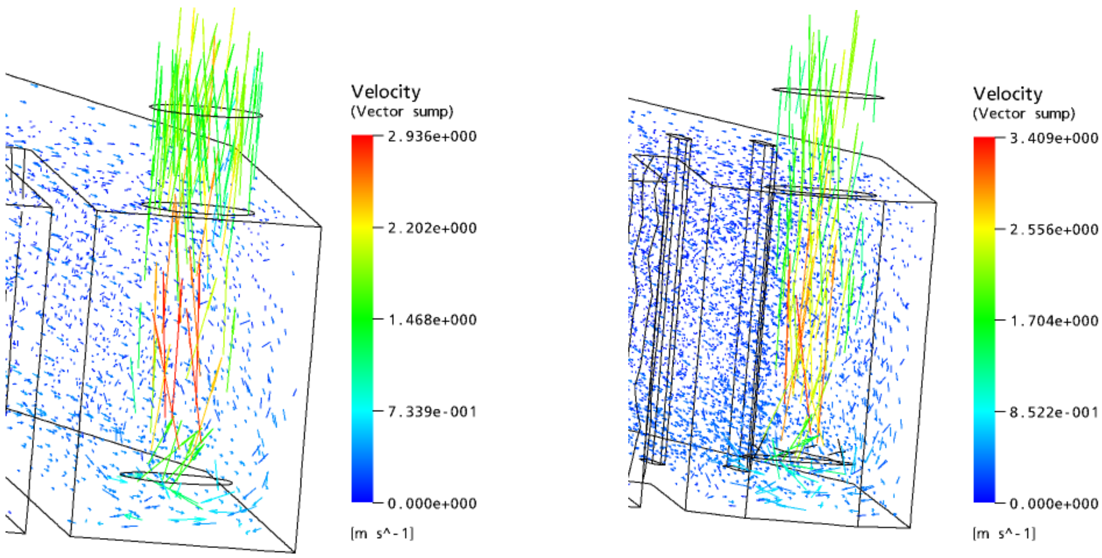


At free surface



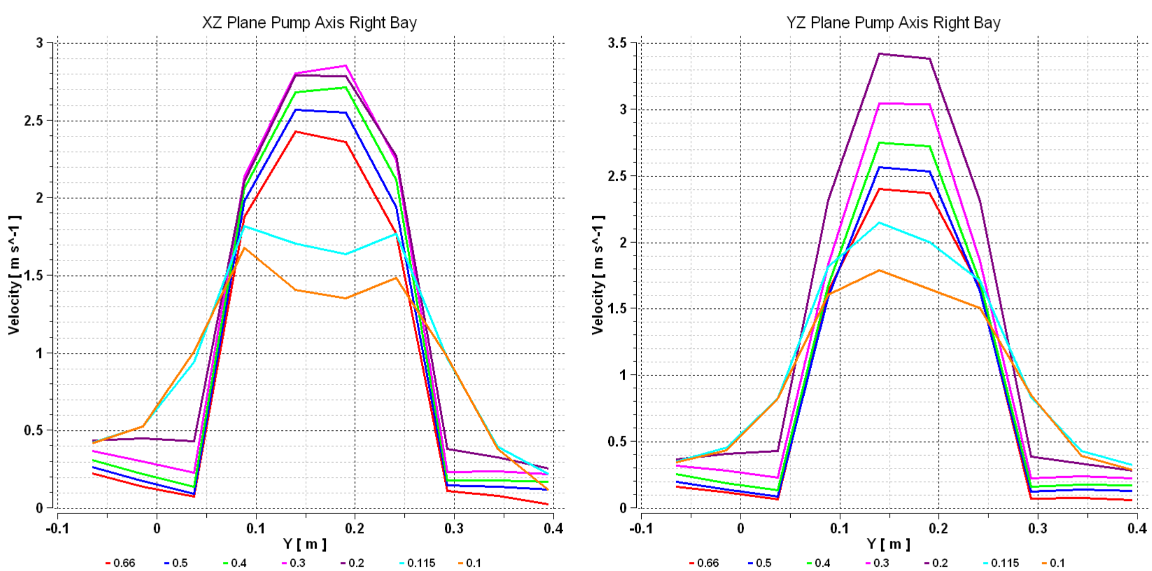
At 0.4m from channel bottom

Figure 5. ***Streamline patterns in Horizontal planes in final model***



Original Model *Final Model*

Figure 6. ***Velocity Vector Plots in right bay***



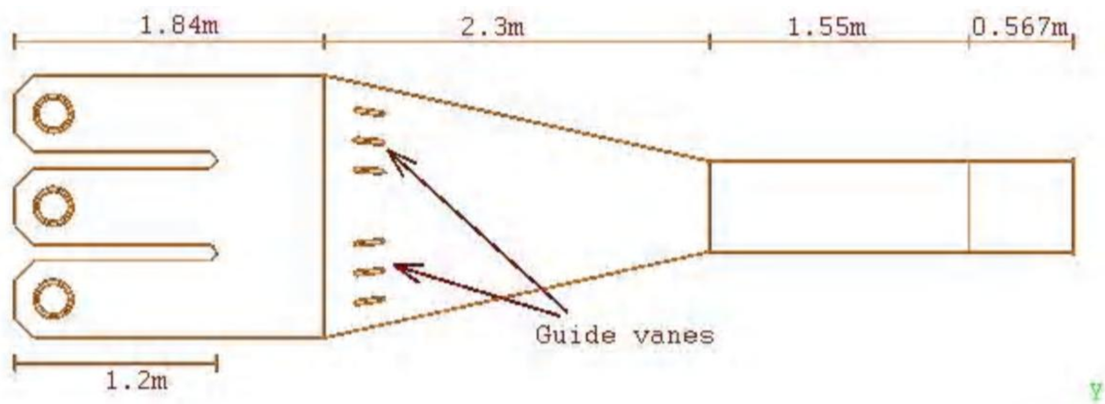
Original Model *Final Model*

Figure 7. ***Velocity profiles across the pump axis in right Bay***

A comparison of the velocity vector plots inside the pump column (Figure 6) also show that the velocities have increased at the inlet as well as in the bellmouth section in the final variant of the model, however the swirl has reduced. The velocity profiles (Figure 7) in the end bay show that, the velocity distribution both at the inlet to the pump column as well as inside the pump column has become much uniform which is due to reduction in the swirl. In the final variant, the magnitude of velocities inside the pump column show an increase of about 20-22% upto a distance of 2.6D from the inlet section, as compared to those in the original model. Beyond this, the velocities decrease gradually till the free surface level in both the variants, however the velocity at the free surface is about 6% less in the case of final variant.

6. EXPERIMENTAL VALIDATION

A physical model confirming to the configuration (Figure 8) 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 two centrifugal pumps (each of 20 hp rating) 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 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.



Plan

Elevation

Figure 8. ***Geometry of the model used for experimental Investigations***

At the beginning of the experiment water was filled in the sump to the desired depth and the pumps started. The discharge through both the pumps was set to the model discharge (0.137m^3/s) and the flow was allowed to stabilize and become steady. The measures applied, to further minimize the high turbulence are: i) Gunny bags at the end of delivery pipe ii) use of grid and wooden planks in the supply tank. The velocity at the end of forebay is measured by a pitot tube. The measurements of the depth of flow in the approach channel are also made. The variation of depth of flow in approach channel is shown in Figure 9.

A number of tests were conducted at twice the Froudian conditions, for minimum water level. Initially dye injection method was tried for observing the flow pattern. For this, the dye was first injected through the dye injection arrangement provided at 0.5m depth from the sump bottom. It was observed that the dye streaks moved hardly to a distance of about 10cm before the turbulence of the rotating mass dissipated them and they mixed up with water.

Next the dye injection was tried at a depth of 0.3m depth from the sump bottom. Here the dye streaks moved for a slightly greater distance than in the previous case before dissipating.

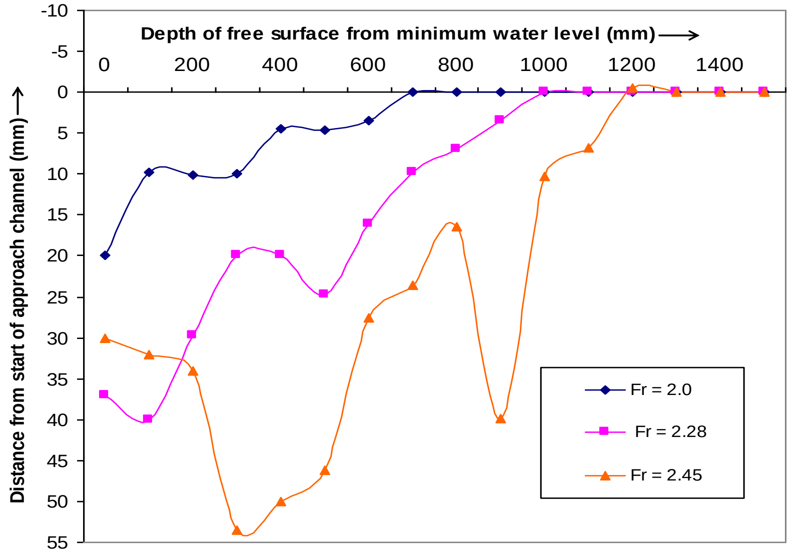
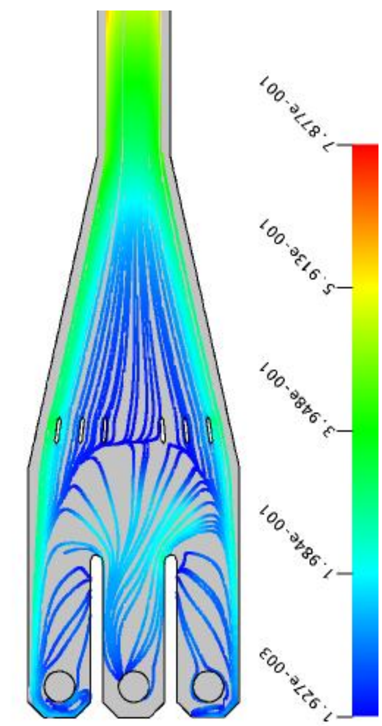


Figure 9. ***Water surface profile***

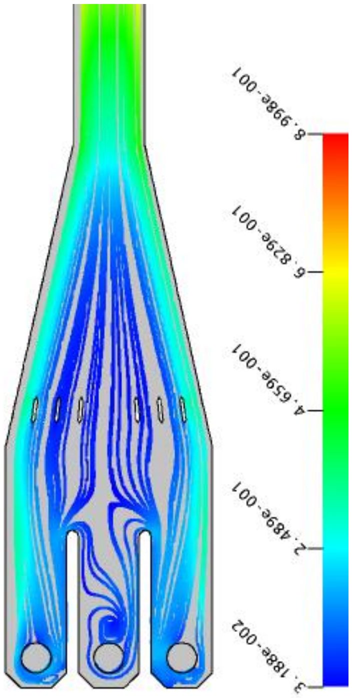


Streamline pattern at Free surface

Figure 10. ***Photographs at free surface***

The pattern of the rotating mass could not be determined with the help of dye injection. Hence the returning flow patterns on the surface were observed and positions of the vortices identified visually. To facilitate the observations light floating material such as thermocol balls and sparkling matter was sprinkled at these positions in order that the flow patterns at these positions may become clearly visible.

The flow conditions observed in the experimental model were in good agreement with the computational results. Experimental observation of the flow pattern formed at the free surface due to returning flows from central bay matched perfectly in shape, size as well as extent with the streamline pattern obtained through computational studies. Photographs of Figure 10 show the progression of the streamlines of returning flow at the free surface theoretically and experimentally. The concentration of flow along the sidewalls as well as in the central bay is also clearly visible in the picture. The flow pattern near the piers, at a depth of 0.4m from the sump bottom can be seen in photographs of Figure 11, which agree well with the streamline pattern obtained through numerical simulation. The computational and experimental results are seen to match closely.



Streamline pattern at 0.4m

Figure 11. ***Photographs at 0.4m from sump bottom***

7. 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.

8. 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