

NUMERICAL PREDICTION OF AIR ENTRAINMENT IN PUMP INTAKES

by

Shyam N. Shukla

Divisional Manager

and

J.T. Kshirsagar

Vice President

Corporate Research & Engineering Division

Kirloskar Brothers Limited

Pune, India



Shyam N. Shukla is working as Divisional Manager in the Corporate Research and Engineering Division of Kirloskar Brothers Limited, in Pune, India. He has been associated with Kirloskar Brothers Limited since September 1996. He has done extensive research in pumps and the pump intake field and has presented papers in Germany, Singapore, Canada, the United Kingdom, and the United States

of America. Mr. Shukla is an extensive user of commercially available CFD software. He has published more than 25 papers in various national and international conferences.

Mr. Shukla graduated (Engineering) from the Government Engineering College, Rewa, India, and he is a post graduate from Maulana Azad College of Technology, Bhopal, India. He started his career in the Problems Oriented Research Laboratory, MANIT Bhopal, India, and then joined the Research and Development (Pumps) division of Jyoti Limited, Vadodara, India.



J.T. Kshirsagar is currently Vice President of the Corporate Research and Engineering Division of Kirloskar Brothers Ltd., in Pune, India. He is also a distinguished Visiting Professor at the Indian Institute of Technology, Madras, India. Dr. Kshirsagar is a Chartered Engineer and Fellow of the Institution of Engineers, India, and a member of ASME, the Indian Society of Hydraulics, and Life Member of the Fluid

Power Fluid Machinery Society.

Dr. Kshirsagar graduated (Engineering) from Visvesvaraya Regional Engineering College, Nagpur, India, and has a Ph.D. degree (Engineering, 1989) from the Indian Institute of Science, in Bangalore, India. He is also a panel member of the Experts Health Monitoring System for Rotating Machinery, Ministry of Information Technology of India.

ABSTRACT

Pump intake is designed to provide adequate water supply to pumps. It is essential to design the pump intake to provide fairly uniform and swirl free flow to pumps. However, it is not always possible due to site constraints. Investigation of pump intake is required prior to installing the pumps at site. There are two

approaches (experimental and numerical) followed for such investigation. The present paper describes a case study wherein air entrainment (vortex with full air-core) was observed in experimental study at lowest water level (LWL) in the original geometry. The pump intake was modified with a goal to eliminate air entrainment. The experimental model was built with these modifications and the air entrainment was seen to be eliminated. The model was investigated using a numerical approach. A commercial code was used to carry out two-phase flow studies to capture air entrainment. The paper brings out results and observations from both numerical and experimental approaches.

INTRODUCTION

The basic purpose of a pump intake is to supply water with uniform velocity at the entry of an impeller. Electric power generating plants utilize circulating-water cooling systems that typically require a number of large-scale pumps to draw water from a river or a reservoir. The fluid flow in pump intakes is rather complex involving expansions and turns together with fluid structure interactions. It is essential to ensure that the pumps operating in such pump intakes get smooth swirl-free flow at their inlets. Proper intake design provides uniform swirl free flow to the pumps. Intakes of such pumps and the geometrical layout of the channel surrounding the pump bells are usually designed in an empirical fashion, relying on laboratory model studies and experiences with previous installations. The Hydraulic Institute Standards specify general guidelines for the design of pump intakes. The site constraints usually call for a deviation from the Standards. It then becomes essential to investigate the pump intake to ensure smooth flow over the entire flow range of the pumps and in all the combinations of the pumps. An experimental investigation on a scaled down model with a scale ratio of about 8 to 20 applying Froude similarity rules typically gives clues to the suitability of the pump intake design. Some cooling system pumps are known to experience certain common operational problems, such as vibrations, impeller damage due to cavitations, and excessive bearing wear resulting in severe deterioration in performance and significant increase in operational and maintenance cost. These problems are associated with certain undesirable characteristics of the flow field in the vicinity of the pumps and are caused primarily by poor design of the intake or channel surrounding the pump bell, or insufficient pump intake submergence depth. Poor pump intake design, for instance, can result in approach flow with high swirl levels, while inappropriate geometrical layout near the pump bell may lead to strong boundary-attached surface vortices. This nonuniformity in the intake flow promotes vibrations and excessive bearing loads. Low pump intake submergence depth could result in formation

of the air entraining free surface vortices, a phenomenon that significantly complicates the flow field and promotes cavitations. Sediment deposition and ingestion cause additional operating difficulties and demand frequent maintenance.

The vortices that form near the pump bell may be classified as free surface vortices (which could transform into air-entrainment) and wall attached vortices (submerged vortices). The usual solution is to conduct an experimental study on a reduced scale model to identify the source of the problem and find a practical solution to rectify them. Although factors affecting the formation of vortices at pump intakes have been known in general terms for quite some time, there is no theoretical method for predicting their occurrence. From a purely numerical perspective, the complexity of the physics is such that it demands the full power of modern computational fluid dynamics (CFD) to solve the equation of a motion and turbulence model in domains. There are additional difficulties associated with modeling free surface and vortex phenomena, because the physics of which are not fully understood. With the latest version of commercially available software, an effort is made in this direction to predict those complex fluid flow phenomena.

This paper describes the use of commercially available software for predicting the formation of air entrainment and submerged vortex with the help of a case study. In experimental investigation, it was found that at lowest water level (LWL) there was a formation of air entrainment. The formation of air entrainment was eliminated with a modification in the pump chamber. Both the test cases are taken into consideration in the CFD analyses. The paper brings out results and observations from both numerical and experimental approaches, and subsequently compares the results in both the test cases.

MULTIPHASE FLOW STUDIES

Multiphase flow refers to the situation where more than one fluid is present. Each fluid may possess its own flow field, or all fluids may share a common flow field. The pump intake flow is a multiphase, free surface flow where the phases are separated by a distinct interface (free surface). In such case the multiphase flows interphase transfer rate is very large. This results in all fluids sharing a common flow field, as well as other relevant fields such as turbulence. In free surface flows, the water flows under gravity, the phases are completely stratified, and interface is well defined. The volume fractions of the phases are equal to one or zero everywhere except at the phase boundary.

An evaluation of CFD capabilities has to ensure that the different types of errors are identified and, as far as possible, treated separately. It is known from single-phase studies that the quantification and documentation of modeling errors (as in turbulence model, etc.) can only be achieved if the other major sources of errors are reduced below an “acceptable” level. These difficulties will be greatly increased by the inclusion of multiphase physics and unsteady effects. In order to tackle the problem, it is necessary to first define the different types of errors, which can impact a CFD simulation. It is then required to list the most promising strategies in order to reduce or avoid these errors. Based on these strategies, procedures have to be defined, which can be used for the test case simulations.

As already depicted, open channel flow phenomena is a complex multiphase and unsteady flow phenomena. The complexity itself is increased manifold if the multiphase flow physics are included. Moreover the authors’ interest in a CFD simulation is to detect an air core that is harmful for the safe operation of pumps. A steady-state solution can depict the formation of an air core that is continuous (or for that sense, exists in the solution most of the time) and hence not desirable for safe operation of pumps. To reduce the complexity of the analysis, the authors’ have carried out two-phase flow in steady-state condition with a view to detect the air entrainment in the pump intake. Knowing that the actual phenomena are always transient, efforts will be made to resolve the transient phenomena in future.

COMPUTATIONAL MODEL DESCRIPTION

Geometry Generation

The sump geometry chosen to test the CFD model replicated the laboratory model (model scale ratio 1:10). The total size of the pump chamber in feet (water domain) is $3.28 \times 1.31 \times 1.03$ (L \times B \times H). The quantity of water that is flowing through the pump chamber is 165.6 gpm (prototype flow rate—52,393.2 gpm). The suction bell diameter is 0.48 ft, while the diameter of the column pipe is 0.361 ft. The bottom clearance is 0.207 ft, while the back wall clearance is kept as 0.525 ft. The submergence at the LWL was 0.82 ft (the required submergence as per Hydraulic Institute Standards is 1.05 ft). The depth of the curtain wall from free surface level in the modified sump was kept as 0.207 ft and the distance from the back wall was kept as 2.2 ft (Figure 1). The average velocity of flow at the inlet is 2.272 ft/s and at entry of the suction bell, it is 2.034 ft/s. The Froud number calculated at entry of the suction bell is 0.52.

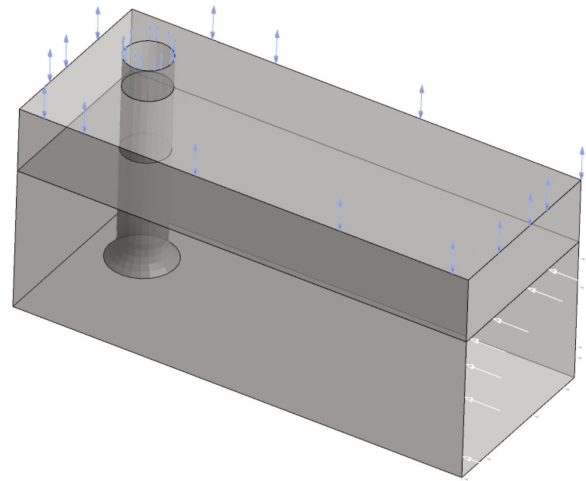


Figure 1. Geometry of Pump Bay (Original Geometry).

As the basic aim is to find the fluid flow phenomena around the pump, only one bay’s geometry was prepared. The geometrical dimensions were kept the same as that of the experimental model. The geometry was divided into various blocks and the grid generated as the structured volume mesh (hexahedral). Distribution of the mesh seeds near the pipe region and near the walls was kept dense as compared to the other region as the velocity gradients were more in that region. The geometry of the pump bay is shown in Figure 1, while the computational mesh of the geometry is shown in Figure 2.

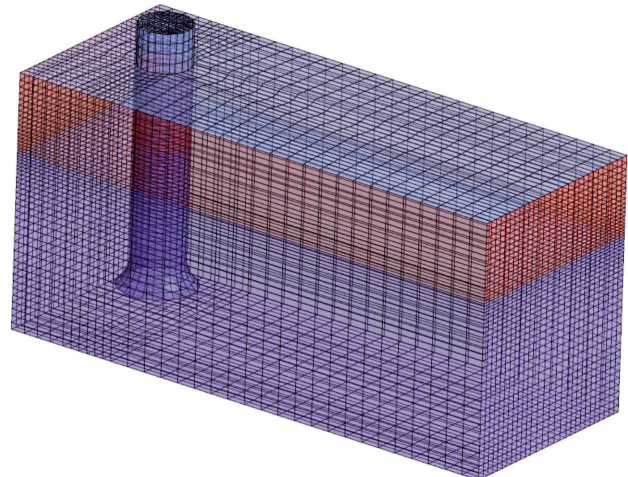


Figure 2. Grid Surface Plot of Pump Bay (Original Geometry).

Boundary and Initial Conditions

The numerical solution of any system demands well-defined boundary conditions. Looking at the complications of the physics of multiphase flow, the following boundary conditions were considered to be appropriate.

The geometry is divided into three main domains of two phases (i.e., water and air) namely pump intake (water phase), pump intake (air phase), and pipe (water phase, representing the pump). A homogeneous model of a multiphase option is selected as the fluid model. The materials for two-phases of fluids are taken as water and air. Both the phases are distinctly defined by virtue of giving initial volume fraction as either one or zero. The volume fraction of water in both water phases is assigned as one and air as zero. The volume fraction of air in the upper air phase is assigned as one and water as zero. Both the fluids are treated as continuous fluids. Initial free surface was the separation of pump intake water domain and pump intake air domain. The quantity of mass flow water was given at the inlet of the pump chamber in water domain. The top of the air domain is assigned as opening with static pressure for entrainment. The outlet of the pipe was also defined as opening. The Cartesian velocity components were assigned to the outlet. The difference in density between phases (water and air) produces a buoyancy force in multiphase flows. Hence buoyant option, with density difference fluid buoyant model, is assigned to all domains. Buoyant reference density of air is assigned in all domains. It is said to choose the density of the lighter fluid since this gives an intuitive interpretation of pressure (i.e., constant in the light fluid and hydrostatic in the heavier fluid). This simplifies pressure initial conditions, pressure boundary conditions, and force calculations in postprocessing. All other surfaces are defined as smooth walls with no slip condition.

The turbulence model, which is used in the solution was a standard $k\epsilon$ turbulence model. The working fluid (water) is modeled as incompressible and the fluid flow state as steady-state.

CFD analysis requires initial flow parameters to initiate the iteration process. This multiphase flow analysis is very sensitive to the initial guess. If the initial guess is proper then the solver gives good results. That is why all the domains are initialized individually. The volume fraction in the pump intake and pipe (water phase) is initialized as 100 percent water, while pump intake (air phase) is initialized as 100 percent air. Other quantities are given as automatic. The discretization scheme used was a high resolution scheme. With this setting, the blend factor values vary throughout the domain based on the local solution field in order to enforce a boundedness criterion. In flow regions with low variable gradients, the blend factor will be close to 1.0 for accuracy. In areas where the gradients change sharply, the blend factor will be closer to 0.0 to prevent overshoots and undershoots and maintain robustness. Here a value of 0.0 is equivalent to using the first order advection scheme and a value of 1.0 uses second order differencing for the advection terms. The advantage of this scheme is that the solution is accurate to the second order while maintaining the robustness.

EXPERIMENTAL MODEL DESCRIPTION

The model sump (at 1:10 scale to prototype) was constructed in brick work and transparent acrylic wherever needed. The back wall portion of the sump was made from transparent acrylic sheets. The model pump consists of a simulated suction bell of transparent acrylic connected to transparent acrylic pipe simulating the internal diameter of column pipe of the prototype pump. A four-bladed swirl meter was placed at the exit of the suction bell of the model pump. The discharge piping from the pipe was connected to a common header and hence to suction of another pump used for recirculating water in the closed circuit. The discharge from each model pump was measured by using a calibrated orifice plate.

The four-bladed swirl meter supported by a low friction-pivoted shaft was located inside the pump column pipe. The revolutions of the swirl meter (i.e., swirl rpm) taken as measure of extent of the vorticity, was counted using a stopwatch, by means of which swirl angle can be determined. Formations of free surface vortices and subsurface vortices were visually observed. Suitable dye was injected for proper visual observation of vortices. The surface flow pattern in forebay and pump chambers was observed by using a wool thread probe/dye injection.

RESULTS AND DISCUSSIONS

The experimental investigation of the original geometry revealed that there was a formation of air entrainment at LWL. Photographs given in Figure 3 and Figure 4 show the location of air entrainment vortices. The sump was modified by bringing down the existing curtain wall below LWL from the previous location (that was above LWL). It was observed that repositioning of the curtain wall helped to eliminate the formation of air entrainment (Figure 5). The swirl angle measured at an approximate location of the impeller eye was nil in both cases. The formation of air entrainment and elimination of same motivated the authors to use of this experimental investigation as a validation case for CFD analysis.

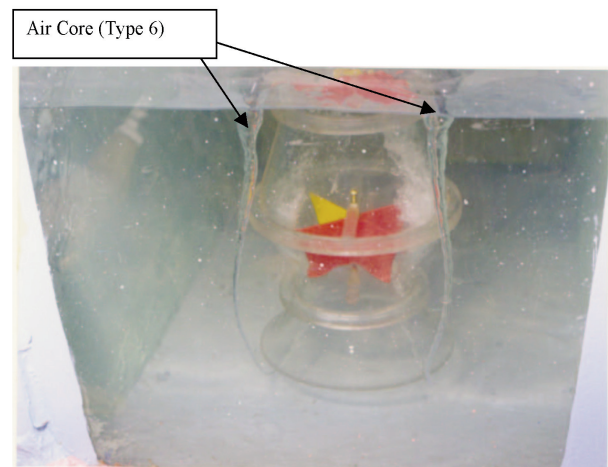


Figure 3. Air Entrainment Seen in the Experimental Sump Model Study A (Original Geometry).

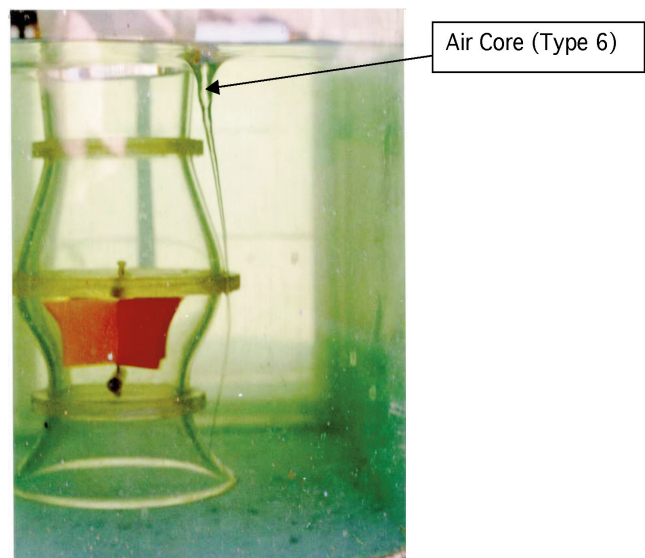


Figure 4. Air Entrainment Seen in the Experimental Sump Model Study B (Original Geometry).

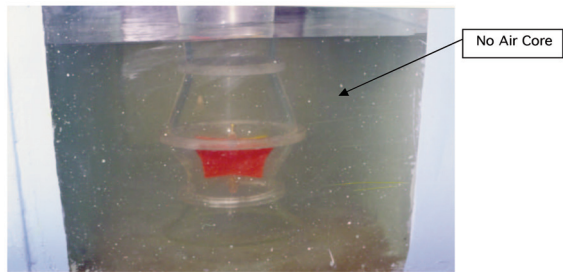
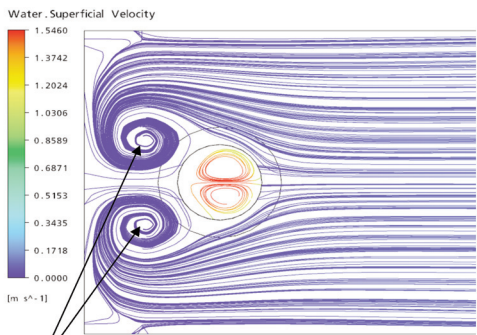


Figure 5. No Air Entrainment Seen in the Experimental Sump Model Study (Modified Geometry).

The CFD analysis results are presented in the form of a streamline plot while photographs of the experimental investigation are attached as validations. The swirl angle in each case is measured at the approximate location of the impeller eye. Additionally the vortex strength at the approximate location of formation of air entrainment is also calculated in the original and modified sump.

One of the ways to gain insight into the flow is to plot the instantaneous “streamlines” using only the velocity components in chosen planes. It should be understood at the outset that, in general, they are neither streamlines nor particle paths, nor their projections onto the plane. They are, however useful devices to identify certain kinematics features of the flow. The “streamline” constructed from the horizontal components of the velocity vector is shown in Figure 6 at the horizontal planes just below free surface. The figure shows the formulations of a large free surface vortex situated between the back wall and the pipe. The symmetry of the vortex field is very evident. The vorticity magnitudes in the cores of these vortices are calculated at a circular plane covering the vortex core just below free surface. The contour plot of vorticity (Figure 7) is found in the range of 0.49 to 0.86 radians/s, which is very high. This high vorticity corresponds to average swirl angle as high as 48 degrees. The foregoing figures still do not convey any impression about the three-dimensional structures of the vortices themselves. The three-dimensional streamline plot of Figure 8 shows the structure of the air core in the water phase clearly and its entry into the pipe. Figure 8 shows the complex vortex pattern formed by smooth approaching flow when interrupted by an obstacle in a confined region. The leading edge of the pipe shows symmetrical lines of separation due to an adverse pressure gradient. The flow acceleration over the pipe and subsequent flow separation is very well captured. The Von-Karman vortex-like flow structure behind the pipe is the result of confining the flow by rear wall. The location of the air core that is found in CFD analysis is well matched with the photograph taken during the experimental investigation (Figure 3 and Figure 4). It is worth noting the closeness of flow patterns obtained from the two different approaches. The air core is fully developed and entering into the pipe (similar to type 6 as defined in Hydraulic Institute standards; Figure 9 is attached for reference).



Fully developed vortices (matches observation by the model study)

Figure 6. Surface Streamline Plot of the Pump Bay Just below Interface (Original Geometry).

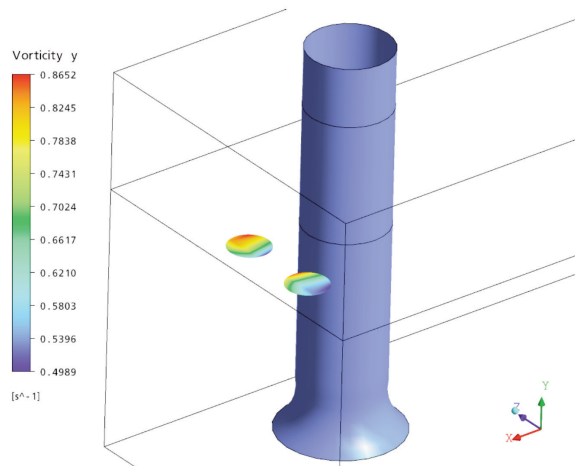


Figure 7. Contour Plot of Vorticity Just below Interface (Original Geometry).

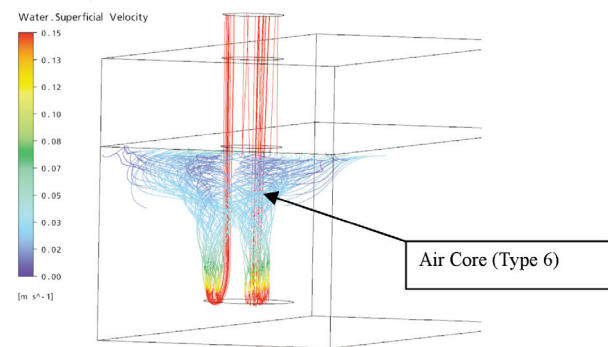


Figure 8. 3-D Streamline Plot of the Pump Bay Showing Air Entrainment (Original Geometry).

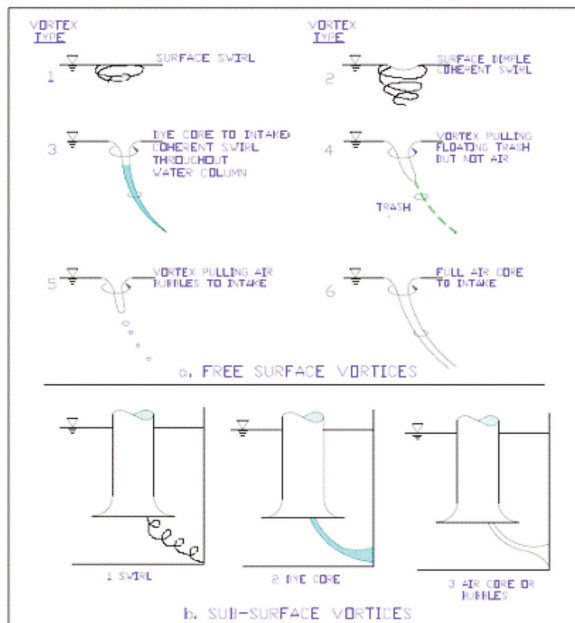


Figure 9. Classification of Free Surface and Subsurface Vortices.

In the original sump configuration, a curtain wall was present above LWL. Opening of the same was reduced and the curtain wall was brought below LWL to prevent air that is flowing along with water to enter into the pump. This modification worked in the experimental investigation. Similar results of the modified sump

are given in Figure 5 and Figures 10 to 12. The same geometry is taken for CFD validation. The “streamline” constructed from the horizontal components of the velocity vector is shown in Figure 11 at the horizontal planes just below free surface. The figure shows the absence of formulations of a large free surface vortex that was present in the original geometry. The vorticity magnitudes in the cores of these vortices are calculated at a circular plane covering the vortex core just below free surface. The contour plot of vorticity (Figure 12) is found in the range of $(-)$ 0.22 to 0.014 radians/s, depicting the weakness of vorticity. This vorticity corresponds to average swirl angle to be as low as 2 degrees. A three-dimensional streamline plot of Figure 10 shows no air core behind the pipe. Here again is seen close resemblance of flow captured by using two different approaches (Figure 5).

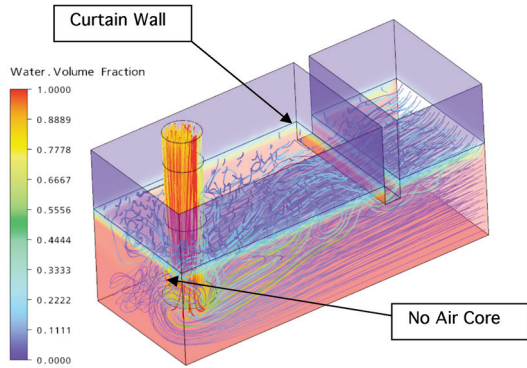


Figure 10. 3-D Streamline Plot of the Pump Bay in Modified Sump (Modified Geometry).

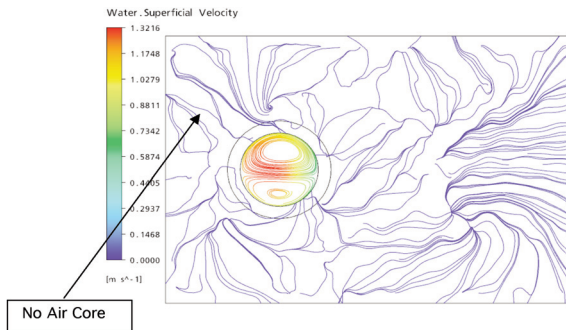


Figure 11. Surface Streamline Plot of the Pump Bay Just below Interface (Modified Geometry).

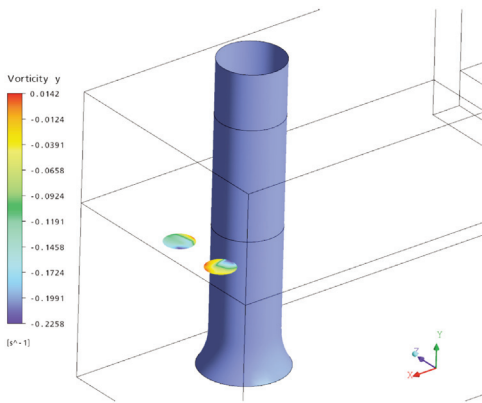


Figure 12. Contour Plot of Vorticity Just below Interface (Modified Geometry).

The quantitative results are compared in the form of swirl angles taken at an approximate location of the impeller eye. The average directional vorticity is calculated at the cross section

selected at an approximate location of the impeller eye. The tangential velocity component of swirl is calculated using vorticity value. The swirl angle is then calculated as the arc tan of tangential velocity to that of axial velocity. These swirl angle values are given in Table 1. The values show the comparison of CFD analysis results to that of experimental investigation. In the experimental investigation, the swirl angle is practically nil in both cases, while in CFD analysis it closely matches the original sump CFD case. Swirl angle calculated in the other case deviates a bit from that of the experimental investigation.

Table 1. Swirl Angle Values.

Sr. No.	Test Case	Swirl Angles in Degrees	
		CFD analysis	Experimental
1.	Original Geometry	0.03	0
2.	Modified Geometry	-2.03	0

CONCLUSION

This paper describes a case study wherein during experimental investigation, formation of air entrainment was observed at LWL and the same was rectified by modifying the height of the existing curtain wall. Both the test cases are taken for the CFD analyses. A commercial code is used to carry out two-phase flow studies to capture air entrainment. This paper brings out results and observations from both numerical and experimental approaches and subsequently compares the results. The solution of the flow in the model intake with a single circular pipe withdrawing water from a rectangular channel illustrates the complexity of the flow and the development of the vortices. The CFD model predicts the flow in sufficient details to identify the locations, size, and the strength of the vortices. In the original geometry, the air entrainment and its location are well captured in the analysis. The results are in accordance with the experimental investigation. The CFD analysis of modified geometry shows the absence of an air core as was expected from the test results. The flow pattern of CFD analysis is matched well with the experimental results.

The model is used to demonstrate the capability of a commercially available code’s multiphase capability. However further systematic studies are needed to understand the effects of various flow and geometric parameters on vortex formations that are encountered in some pump intakes. Improvements in the treatments of the free surface boundary conditions should be considered along with these studies.

BIBLIOGRAPHY

ANSI/HI 9.8, 1998, “American National Standard for Pump Intake Design,” Hydraulic Institute, Parsippany, New Jersey.

ANSYS—CFX Users Guide, 11.0, M/s ANSYS Inc., USA.

Constantinescu, G. S. and Patel, V. C., February 1998, “Numerical Model for Simulation of Pump-Intake Flow and Vortices,” *Journal of Hydraulic Engineering*.

Kshirsagar, J. T. and Joshi, S. G., 2003, “Investigation of Air Entrainment—A Numerical Approach,” Fluid Engineering Division’s Summer Meeting 2003—45415, Proceedings of the Fourth ASME-JSME Joint Fluid Engineering Conference, Honolulu, Hawaii.

ACKNOWLEDGEMENTS

Authors would like to place on record their gratitude to the management of Kirloskar Brothers Ltd. Pune, India, for the encouraging attitude toward the Research & Engineering Division, which led to this paper. They are also very thankful to their colleagues in the Research & Engineering Division, Pune, for their cooperation during this work. The authors’ would like to thank Mr. Michael Cugal, their monitor, for prompt response and keen review of the paper.

