International Journal of Engineering Research and Technology. ISSN 0974-3154 Volume 10, Number 1 (2017)
© International Research Publication House http://www.irphouse.com

# CFD ANALYSIS OF FLOW IN PUMP SUMP AND PHYSICAL VALIDATION FOR BETTER PERFORMANCE OF PUMP

*Pravin Kumbhar[1*], P.B.Patole.[2], HaribhauMohite[3]*

*[1]M.E Student, BharatiVidypeeth College of Engineering, Kolhapur, India*

*[2]AssociateProfessor, BharatiVidypeeth College of Engineering, Kolhapur, India*

*[3]Associate Manager, Kirloskar Brothers Limited Kirloskarvadi, India*

*[*]pravinkumbhar05@gmail.com*

---

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.

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

---

## 1. 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 suppressiondevices(Cruciform), flowstraightner, 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

## 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.A Hydraulic Institute standard 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, corner 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.

## 3. Geometry of Computational Model

Computational study was conducted for the pumping system of a cooling tower having 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 1:10 was 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.

The ANSYS CFX 15.0 Solver module of ANSYS CFX-15.0 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.

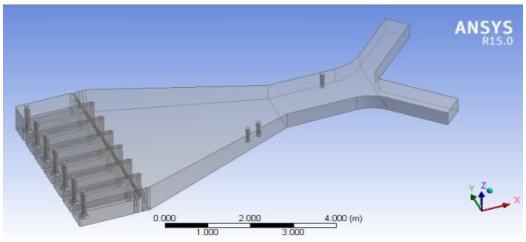


**Fig 2.Modelled basic sump 1 geometry.**

Figure 2 SHOWS 3-D model of sump which is created in Solid Works.

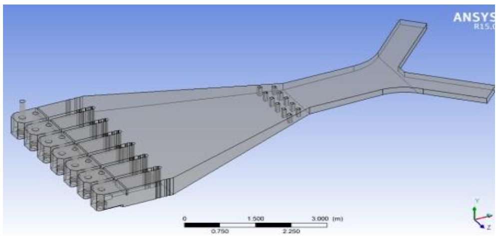


Fig-4: Modelled basic sump 3 geometry.

### *3.1Sump 1*

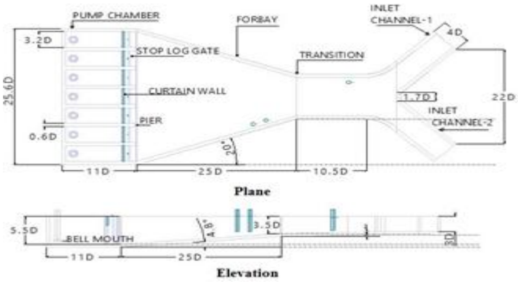


**Fig-1.Model with dimensions (a). Elevated view of model (b) Top view of Model**

Fig-1 gives the schematic diagram of the model in plan and elevation showing all the basic dimensions.

### *3.2 Sump-2.3*

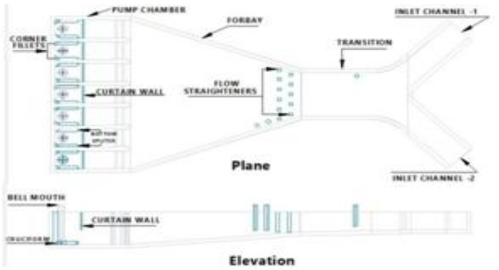


**Fig-3.Modelwith all modification (a).Elevated views of model (b) Top view of Model.**

In this above Fig sump-2 and sump-3 common diagram.In Sump-2 required all modification such as cruciform, Bottom Spliters, corner Fillets and flow straighteners. In sump -2 at the testing time surface vortices and air entering at low intensity as compared to sump-1.due to this reason, In sump-3 some design changes, Pump Sump-3 are done according to HI (Hydraulic Institute) standard or ASME Criteria for a pump sump design.

The computationa investigations were performed using ANSYS CFX 15.0. The inputs and outputs of both the software's 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 which is shown in figure 3 In general the mesh generated for different variants had about 17 to 20 lakhs elements with the number of nodes varying from 3 to 4 lakhs.

Meshing nodes and element are as follows

**Table 1: Comparisons of Nodes and Elements.**

| Model | Nodes | Elements |
|---|---|---|
| Sump 1 | 336344 | 1758929 |
| Sump 2 | 389576 | 2022602 |
| Sump 3 | 212868 | 1096281 |

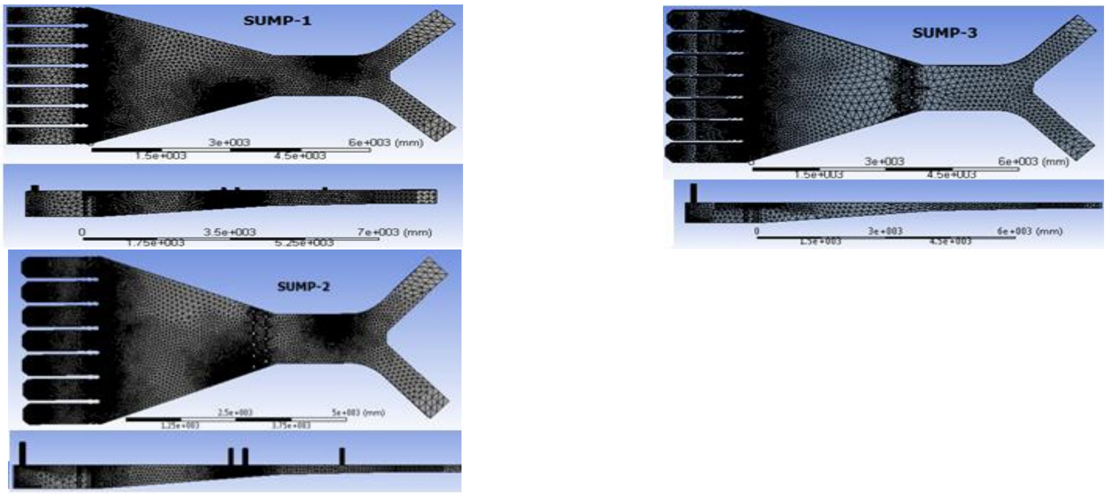


Fig-5: Meshing Analysis.

The solver control parameters were specified in the form of solution scheme and convergence criteria.

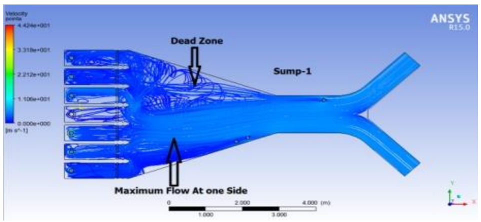
**Table 2: CFX Solution Setting**

| Sr. No | Setting | choice |
|---|---|---|
| 1 | Simulation | 3D |
| 2 | Solver | Pressure Based |
| 3 | Model | The turbulence model was selected as K-€ model. |
| 4 | Material | Water |
| | Morphology | Continuous Fluid. |
| 5 | Pressure Velocity Coupling scheme | Simple |
| 6 | Reference Pressure | 1atm |
| 7 | Heat Transfer Model | Isothermal |
| 8 | Fluid Temperature | 25° |
| 9 | Flow Regime | Sub Sonic |
| 10 | Mass And Momentum | 5m/s |
| 11 | Turbulance | Medium Intensity and eddy Viscosity Ratio. |
| 12 | Gradient | Green-Gauss Node Based |
| 13 | Discretization Pressure | Standard |
| 14 | Turbulent Kinetic Energy | First Order Upwind |
| 15 | Turbulent Dissipation Rate | First Order Upwind |
| 16 | Discretization Momentum | Second Order Upwind |
| 17 | Compute from | Outlet |

## 4. 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. Maximum Flow At one side and another side of sump is dead zone Create.

### *4.1SUMP*-1



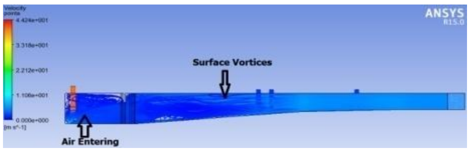


Fig-6 Streamline pattern in Sump-1

Fig-7 Photographic view showing air entering.

### *4.2SUMP-2*

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-2 geometry such as cruciform, corner fillet, bottomspliters and flow straightneretc Shown in fig-3,Inspite of the various modifications in the sump-2 geometry and provision.However each modification was aimed at reducing Air entering, Swirl angle, surface vortices the extent of the rotating mass in the central bay and thus making the flow conditions in the forbay is more uniform.

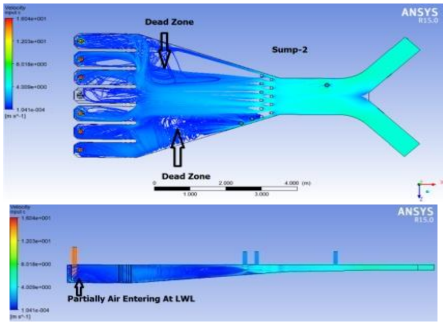


Fig-8: Streamline pattern in sump-2 Model.

The results of the Sump-2 selected configuration showed improved flow conditions from amongst all the variants. For the sump-2 model the streamline pattern is continuous as compared to sump-1 but defects are not remove totally in sump 2 the intensity of defects is low such as partially air entering,and Partially dead zone created on sump-2 shown in fig-6.

## 5. Experimental Validation

The experimental setup was fabricated as a recirculating system with water from the pump bays in the sump being pumped by seven centrifugal pumps to the stilling tank. For discharge measurement, sharp edged orifice meters with d/D ratio 0.67 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. Acrylic windows were provided in the sidewalls and backwall (one in each pump bay) of the sump to facilitate visual observations.

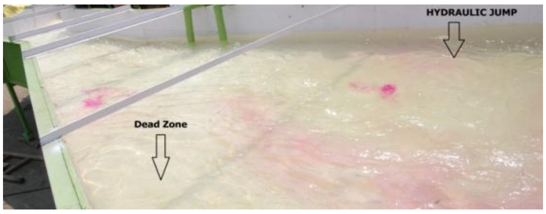


Fig-9 Photographic view showing hydraulic jump and dead zone.

Fig-10. Geometry of the model used for experimental Investigations.

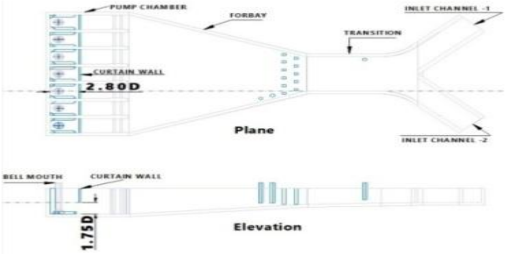


Fig10: geometry of model

Fig-11: Location of curtain wall

### *5.1 Sump-3*

Pump Sump 3 with All Modification Only Changing the Position of Curtain Wall (Curtain Wall Shift 640mm from Corner Fillet and 549mm depth from the sump bottom) due to this change remove all defects from sump. And also reduce the swirl angle as shown in fig-9.

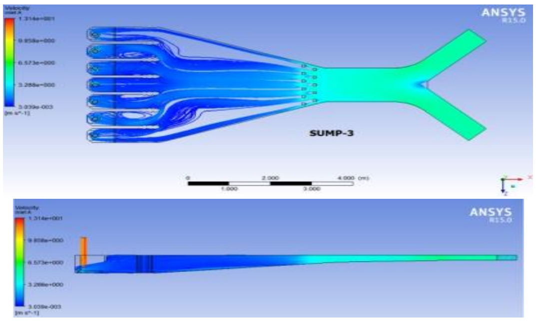


Fig -12: Streamline pattern at Free surface.

For the final model of sump-3 the streamline pattern is continuous throughout the sump. Compression of streamlines towards the sump-1, sump-2and sump-3.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 not observed of the air entering, surface vortices, and reduce the swirl angle identified visually.

*5.2Swirl and Swirl Angle Estimation.*

A quantifiable index of the vortex activity entering the intakes is the swirl angle measured in the riser pipe downstream of the bell mouth throat. The revolutions per unit time of the swirl meter are used to calculate the swirl angle, which is indicative of the intensity of flow rotation. The angular velocity of the vortometer is measured by counting the revolutions over a period of 60 seconds. The tangential velocity is then obtained. This velocity is divided by the axial velocity obtained by dividing the measured flow by the area of the riser pipe. The swirl angle is the arc tan of this ratio.

**Table 3: Flow Condition and Rates**

| Flow Condition | Model Flow |
|---|---|
| 1.0F | 22.18l/s |
| 1.5F | 33.27l/s |
| 2.0F | 44.36l/s |

Swirl angle, θ = 5°.

$$\mathrm{Tan}\theta = \frac{U}{V} \qquad (1)$$

Here U and V are the angular velocity and the radius of the vortometer and is the average axial velocity in the pipe. The physical principle used in the estimation of the bulk swirl angle in the present work is that in the absence of torque the angular momentum in a given direction will be preserved. As there will be distribution of angular momentum, the total angular momentum P can be computed as

$$U = \frac{\pi \times D \times N}{60} \qquad (2)$$

The bulk angular velocity can be obtained by the equation,

$$\text{Velocity at bell inlet (V)} = {}^{Q}\!/_{A} \qquad (3)$$

Sample Calculation for Swirl RPM CCW pump (BHQ 95D)

Diameter at exit of Bell Mouth= 107mm.

Vortometer diameter, D =80 mm.

N = RPM of vortometer.

Flow condition = 1.0F.

Model flow (Q) = 22.18 l/s

(Corresponding to prototype flow=25250$m^3$/hr)

$$= 0.02218\ \mathrm{m^3/s}$$

V θ

**Table 4: Swirl RPM at different Angle.**

$$U = \frac{\pi \times D \times N}{60}$$

$$U = 0.00419 \times N\ \mathrm{m/s}$$

$$\text{Velocity at bell inlet (V)} = {}^{Q}\!/_{A}$$

$$A = \frac{\pi}{4} \times D^2$$

$$A = 0.00899$$

$$V = 2.466\ \mathrm{m/s}$$

$$\mathrm{Tan}\theta = \frac{U}{V}$$

$$N = 51.49\ \mathrm{rpm}$$

**SWIRL RPM FOR DUTY POINT FLOW OF 25250 $M^3$/Hr**

| Flow Condition | Model Flow L/S | Swirl Rpm At 5° | Swirl Rpm At 4° | Swirl Rpm At 3° | Swirl Rpm At 2° | Swirl Rpm At 1° |
|---|---|---|---|---|---|---|
| 1.0 F | 22.18 | 51 | 41 | 35 | 21 | 10 |
| 1.5 F | 33.27 | 77 | 62 | 46 | 31 | 15 |
| 2.0 F | 44.36 | 103 | 82 | 62 | 41 | 20 |

### *5.3 Effectiveness of vortex suppression devices in reducing swirl angle.*

The swirl angle reported in the tables 3 show that the maximum swirl angle is 6.365°. This is close to the maximum swirl angle limit of 5° suggested by Hydraulic Institute [2]. Vortex suppression device can be used to suppress the swirling motion in the pump suction. The vortex breaker designed based on [2] is shown in figure 9 is used in the present work. Experiments and simulations were run with the vortex breaker placed coaxially in the pump centerline. The swirl angles obtained after implementation of the vortex breakers is summarised in table 4. It can be seen from table 4 that the vortex breaker has reduced the swirl angles.

### *5.3.1Cruciform use to Vortex Breaker*

Fig-13: Cruciform.

Fig- 14: Cruciform arrangement.

### *6.4.2Swirl angle measurement.*

Swirl angle, θ=?
N=35 From table-1

$$U = \frac{\pi \times D \times N}{60}$$

$$U = 0.00419 \times 35\,\mathrm{m/s}$$

$$\text{Velocity at bell inlet } (V) = {}^{Q}\!/_{A}$$

$$A = 0.00899$$

Swirl angle measurement as follows,

$$V = \frac{Q}{A} = 2.466\,\mathrm{m/s}$$

$$\mathrm{Tan}\theta = \frac{U}{V}$$

$$\mathrm{Tan}\theta = 0.0594$$

$$\theta = 3.40°$$

## 6. Results and discussion

Swirl angle reading in sump 1 will be higher than acceptable limit Shown in table no.in sump 2and sump swirl angles are found within acceptable limit of 5°as per hydraulic institute standard (HIS),gradually decreased the swirl angle in sump 1,sump2,and sump 3 respectively. Shown in table no. Swirl angle decrease means flow in bell mouth is uniform.when flow is uniform automatically improve the efficiency of pump. Also pump life increased. Shown in table 3, 4 and 5. Swirl angles computed CFD Result Are As Follows,

### *6.1.1SUMP-1*

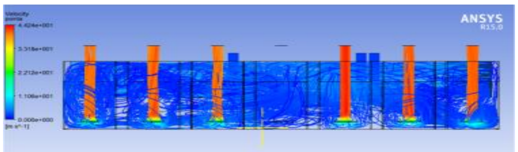


**Fig15-Sump1**

### *6.2.2SUMP-2*

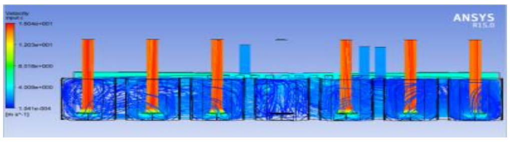


**Fig16-Sump-2**

### *6.1.3SUMP-3*

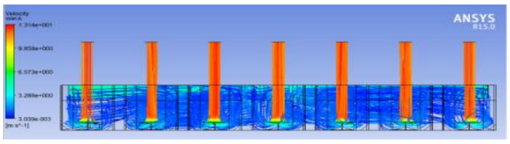


**Fig17-Sump-3**

### *6.2 Swirl Angles Comparision.*

### *6.2.1Sump 1 Geometry.*

| combination | P-1 | P-2 | P-3 | P-4 | P-5 | P-6 | P-7 |
|---|---|---|---|---|---|---|---|
| angles | 8.6 | 2.9 | -6.41 | X | -1.84 | -6.93 | 6.33 |

**Table 5**

| combination | P-1 | P-2 | P-3 | P-4 | P-5 | P-6 | P-7 |
|---|---|---|---|---|---|---|---|
| angles | 3.41 | 2.95 | 2.30 | X | 0.30 | 2.49 | 4.35 |

### *6.2.2Sump 1 Geometry.*

**Table 6**

| combination | P-1 | P-2 | P-3 | P-4 | P-5 | P-6 | P-7 |
|---|---|---|---|---|---|---|---|
| angles | 1.15 | 0.46 | 0.26 | 2.8 | -0.26 | 0.2 | 3.4 |

### *6.2.3Sump 2 Geometry.*

**Table 7**

## 7. Conclusions

The CFD package ANSYS CFX-15 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. A new method has been presented to calculate swirl angles from the velocity field obtained from CFD to a single value which is consistent with the HI standard method [2]. Comparisons have been made between the swirl angle calculation methodology and experimentally obtained values and it is concluded that the numerical calculation method compares well. The effectiveness of the vortex suppression devices has also been demonstrated and serves as a swirl angel check. 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. Acknowledgement:

I would like to thanks **Kirloskar Brother's Limited, Kirloskarvadi** for allowing me to use ASME standards and Books from technical library.

## 9. References:

[1].G. S. Constantinescu, V. C. Patel, Role of "Turbulence Model in Prediction of Pump Bay Vortices", Journal of Hydraulic Engineering, May 2000, pp.387- 391.

[2]Tomoyoshi Okamura, KyojiKamemoto and Junmatsui "CFD Prediction and Model Experiment on Suction Vertices in Suction Pump'', Ninth Asian International Conference on Fluid Machinery October 2007 Jeju, Koria.pp16-1

[3].TanweerS.DeshmukhandV.KGahot"SimulationofFlow through a Pump Sump and its Validation", IJRRAS4, July, 2010.pp387-390.

[4].PeetakMitra, NiranjanGudibande, KannanIyerand TI Eldho "Algorithm for Estimating Swirl Angles In Multi-Intake Hydraulic Intakes'',Department of Mechanical and Aerospace Engineering, Syracuse University, Syracuse, NY, USA,2011.pp28-31

[5].ANSI/HI-9.8, 2012 Rotodynamic pumps for pump intake design.