ISSN: 1687-1006 p
ISSN: 2682-3640 e

The Egyptian International Journal of Engineering Sciences and Technology

https://eijest.journals.ekb.eg/

Vol. 52 (2025) 34–54

DOI:10.21608/eijest.2025.359193.1324

Numerical Study of Controlling Geometrical Parameters of Flow Pattern in Pump Sumps: Effects on Vortices and Water Stagnation

Tarek M. Salah EL-Din[a] , Mohamed Y. Anter[b],

[a]*Lecturer, Irrigation and Hydraulics Department, Faculty of Engineering, Cairo University, Giza, Egypt*
[b]*Ph,D candidate, Irrigation and Hydraulics Department, Faculty of Engineering, Cairo University, Giza, Egypt*

ARTICLE INFO

Article history:

Received 09 February 2025
Received in revised form 03 March 2025
Accepted 03 March 2025
Available online 03 March 2025

Keywords:

CFD
Flow field
Swirl

ABSTRACT

Pump sumps often face issues like water stagnation and vortex formation due to inadequate consideration of hydraulic conditions during design. This research employs Computational Fluid Dynamics (CFD) simulations to explore how different sump shapes and configurations impact flow patterns. Key factors examined include sump geometry, suction direction, inlet level, and the positional relationship between inlet and outlet pipes.

The results demonstrate that circular sumps, high-level inlets, and horizontal suction pipes significantly reduce water stagnation and vortex intensity. Placing inlet and suction pipes on opposite sides further improves flow patterns. These findings provide practical guidelines for designing more efficient pump sumps, enhancing hydraulic performance, and reducing maintenance needs.

Additionally, the study underscores the importance of optimizing sump configurations for various operational scenarios. By integrating advanced computational techniques with robust design principles, the research offers actionable recommendations that can be directly applied to improve the performance and reliability of pump sumps. The insights gained from CFD simulations pave the way for more efficient water management practices, ultimately contributing to reduced operational costs and enhanced system reliability in hydraulic engineering applications.

This study highlights the critical role of detailed hydraulic analysis in pump sump design and demonstrates the value of CFD simulations in developing effective solutions for minimizing common hydraulic issues.

1. Introduction

The sump is a depressed structure where a liquid (usually water) is collected and prepared to start processing or moving between two points through pipes by using pumps. Sumps may have many shapes such as circular or rectangular shapes. Inside the sump, some structures may exist like spillways, wires, and walls. These structures play a role in flow control and formation of the flow pattern inside the sump. The flow pattern inside the sump can highly affect the performance of the pumps. Due to sump geometry, sump inlets and outlets and operating alternatives, generated flow pattern inside the sump may suffer from some defects: vortices and stagnant water body.

1.1. Vortices

Depending on the previous experimental work and practical engineering experience and practice of pump sump geometry selection and dimensioning, several recommendations and layout guidelines have been developed over time to help engineers to layout pump sumps and to avoid the existence of swirls and vortices and other flow pattern issues. Vortices and swirls are one of the major problems that affect the pumps operation and performance. The generated high speed turbulent flow inside the sump develops swirls and vortices as it moves randomly inside the pump sump. The effect of vortices on flow pattern varies depending on their location and intensity.

The vortex and swirl are movement of the fluid particles in a rotational pattern around a stagnant or moving axis. This axis can be either straight or curved starting from either the water surface or sump walls. The particles movement pattern may be used to classify the vortex. The vortex type can be classified according to location and intensity. The main two categories of vortices are free surface and sub-surface types. As shown in Figure (1), there are six types of free surface vortices from type 1 to type 6 according to their intensity. Type 1 represents the lowest intense

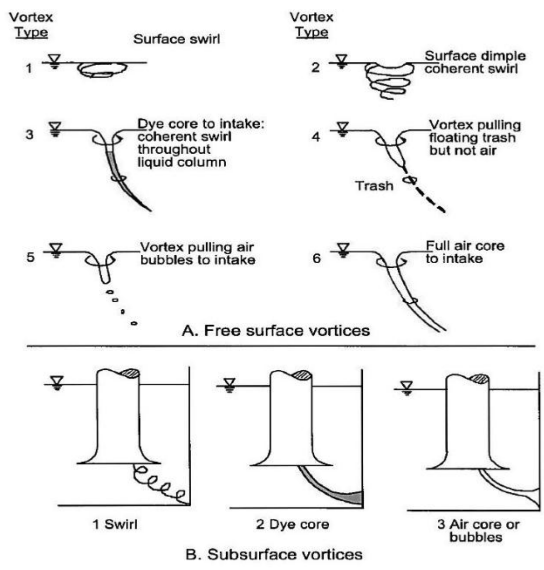

Fig.1: Free surface and sub-surface vortices classification [25]

vortex and type 6 represents the most intense vortex. When the intensity of vortex increases, it entraps more air inside the water. In type 6 vortices, a full cone of air is developed inside intake and air finds its way to the suction pipe.

Vortices formation in pump sump can cause major problems to the pumps and decrease their performance. Pulled air by vortices inside the water body towards inside the pump causes cavitation inside the suction pipe and inside the pump impeller, leading to critical vibrations which cause more wear and damage to the inner pump components and increase the required maintenance and rehabilitation. Vortices also pull floating trash from the surface to inside water. This increases the chance for blocking the suction pipe or damage the pumps impeller. There are several techniques to identify and observe vortex formations in sumps: using dye near the expected vortex formation location, using swirl meter installed in suction pipes, cross-sectional velocity profile, or just using visual observation for free surface vortex types.

1.2. Water Stagnation

The second main problem that can occur to the flow field in the pump sump and can highly affect pumps performance and operation is water stagnation. Stagnation is defined that one or more volume of water is trapped inside the sump and not participating in flow movement in the sump. The stagnant water bodies can be identified inside the sump by profiling velocity on different cross sections in the sump, and seeking the volumes with very low velocity unlike the rest of the fluid volume. Having stagnant water bodies inside the sump reduces the water quality and leads to a higher sedimentation rate in the sump, and results in more need for sump cleaning and maintenance. The water stagnation problem can be a result of poorly designed pump sump, bad pumps arrangement, or unexpected operating conditions.

2. Problem Definition and Study Objectives

2.1. Problem Definition

Pump sumps are major and important component of the water transmission and pumping systems. Flow pattern inside the pump sump suffers from the previously described three major problems: stagnant water bodies in some parts of the sump, strong vortices and swirls. These problems greatly affect the flow field inside the sump and consequently pumps operation and performance and hinder the ability of the sump to perform its function and to transport the required flow with the required head.

Sump shape is usually defined from common practice and its geometry and dimensions are defined from the dimension of the pumps and their spacing without paying attention to the anticipated associated flow problems. The flow pattern inside the pump sump is dominated and affected by the sump shape and its geometry and dimensions. Usually from an economic perspective, the sump is planned as small as possible to minimize footprint and construction cost without paying attention to the flow problems.

Pump sump should be hydraulically planned and dimensioned to achieve smooth and non-disturbed flow field pattern inside the pump sump minimizing or eliminating the previously mentioned common defects and problems in the flow patten. The design process of sump usually includes a set of parameters that control the flow in the pump sump such as shape of the sump, distance between suction pipes, submergence of suction pipes, geometry of the suction bell-mouth, existence of anti-vortex device, width of the pump bay, etc. Studies for best selection of pump sump shape, geometry, dimensions and other factors can be performed through laboratory experiments, field-scale trials or through numerical solution and simulations.

Scaled laboratory physical models of the pump sump with several predefined hypotheticals predicted operation scenarios and configurations are both time and resources consuming, besides, the experiments may not cover all required operating conditions. Based on previous scaled laboratory experiments and field experience, several layout recommendations and guidelines have been proposed and developed for the pump sump. However, these recommendations cannot solve all types, shapes, and configurations of the pump sump and do not solve all anticipated problems in the flow field. In some incidents, a critical case or cases maybe defined to be experimentally modelled separately for precise and accurate results.

Solving the flow field using Computational Flow Dynamics CFD numerical simulations presents robust tool for studying and defining the problems and drawbacks of the flow field for many domain shapes and geometries, operational scenarios and boundary conditions. Analysis of CFD simulations results can present precise definition and selection of suitable sump shape and arrangement, geometry and dimensions for best flow field pattern avoiding previously described problems associated with the flow field in pump sump or minimizing them. CFD can be used to replace high cost and time-consuming laboratorial experimental investigations however, calibration and verification using some experimental results are still needed. CFD simulation is cost and time effective tool that replaces numerous numbers of high-cost and time-consuming laboratorial investigations and leads to optimum pump sump design. High resolution grid used in the CFD simulation produces more precise results. CFD simulations presents numerous results and data that can be analyzed for better understanding flow field details and better suggestions for the required modifications.

2.2. Study Objectives and Methodology

The main objective of the present study is to develop systematic guidelines and recommendations for best selection of the pump sump shape; arrangement; dimensions and other geometrical features to eliminate or minimize the previously described defects and drawbacks associated with the flow field in the pump sump.

CFD simulations using ANSYS 2023 R1 shall be performed using previous common practice and experimental knowledge of pump sump shape and geometry, layout and arrangement. The effect of each parameter on the flow field pattern shall be investigated through numerous hypothetical combinations of geometrical, arrangement, input conditions and operational scenarios. CFD simulation results especially those of the flow field shall be analyzed to develop the required guidelines and recommendations for best selection of sump shape, arrangement and design parameters for elimination of or reducing of the flow field problems.

The methodology used in the present study to develop the required guidelines and recommendations using CFD simulations can be summarized in the following steps:

1. CFD model development and definition of all parameters.
2. Model calibration and validation using authenticated data available in literature.
3. Determination of the hypothetical combination scenarios of pump sump shape, geometry and dimension, arrangement, input conditions.
4. CFD simulations for the hypothetical combination scenarios
5. Analysis of the CFD simulation results and formulating the required guidelines and recommendations.

Model verification ensures that the numerical solution of the CFD model is accurate, reliable and can be used for further simulations. This is achieved by comparing the results of the CFD simulation with previous data available in literatures: authenticated numerical results, experimental results, and or field measurements. Verification is an essential step in CFD analysis, as it helps to identify and adjust any uncertainties or modifications in the model formulation, discretization, and or implementation.

3. Previous Studies

3.1. Previous Experimental Studies

Research on flow behaviour in pump sumps has long focused on experimental investigations to understand hydraulic phenomena such as vortex formation and turbulence. Early studies highlighted the limitations and applicability of scaled laboratory models. Padmanabhan and Hecker (1984) [1] examined the effects of large-scale ratios on free-surface vortices, finding that scaling had minimal impact on hydraulic performance, thereby validating laboratory models for predicting vortex behaviour. Similarly, Odgaard and Dlubac (1984) [2] used hydraulic models to explore the influence of upstream geometry on sump design, demonstrating the utility of potential flow theory for performance enhancement.

Subsequent experimental work sought to develop frameworks for analyzing flow dynamics. Hite and Mih (1994) [3] refined equations for tangential vortex velocity, validating them through laboratory tests and confirming their broad applicability. Arboleda and El-Fadel (1996) [4] emphasized the role of upstream geometry, advocating for customized sump designs tailored to specific conditions. Standards such as ANSI/HI 9.8-1998 formalized these findings, offering guidelines for pump intake design, though later reviews by Westende et al. (2015) [5] highlighted the need for improved consistency in measuring swirl and model acceptance criteria.

Detailed experimental investigations into vortex formation have provided key insights. Ansar and Nakato (2001) [6] used Acoustic Doppler Velocimetry (ADV) to measure swirling flows, generating data critical for validating numerical models. Sarkardeh et al. (2010) [7] examined the impact of intake head-wall slopes on vortex dynamics, concluding that steeper slopes reduced vortex intensity but increased instability. Other studies emphasized the role of geometric configurations. Taştan and Yildirim (2013) [8] found that dynamic similarity in scaled models was not always necessary for predicting critical submergence, while Gaskin et al. (2014) [9] demonstrated how turbulence, viscosity, and surface tension strongly influenced vortex behaviour depending on geometry and flow conditions.

Experimental research has also contributed to understanding free-surface vortices. Sun and Liu (2015) [10] used Particle Image Velocimetry (PIV) to analyze vortex properties in cylindrical tanks, introducing dimensionless equations for critical submerged depth and providing new metrics for characterizing vortex behaviour. More recently, studies have focused on design modifications to reduce vortex formation and improve hydraulic efficiency. For instance, Shinde and Arakerimath (2023) [11] proposed hydraulic performance formulations for centrifugal pump sumps, showing that specific configurations could reduce turbulence and enhance energy efficiency.

3.2. Previous Numerical Studies

Numerical modelling has become an indispensable tool for studying hydraulic phenomena in pump sumps, complementing experimental approaches. Early computational efforts, such as those by Constantinescu and Patel (1998), demonstrated the potential of Computational Fluid Dynamics (CFD) to predict vortex behaviour and optimize design parameters. Neary et al. (1999) [12] expanded on this work by developing three-dimensional frameworks for simulating turbulent flows in lateral intakes, achieving accurate predictions with k-ω turbulence models.

Advancements in numerical techniques have enabled the study of increasingly complex hydraulic systems. Sotiropoulos et al. (2005) [13] introduced unsteady RANS (URANS) models capable of simulating intricate flow conditions, including high Reynolds numbers. Tokyay and Constantinescu (2006) [14] validated the superiority of Large Eddy Simulation (LES) in capturing turbulence and flow dynamics, using scaled laboratory data for verification. Nakayama and Hisasue (2010) [15] applied CFD and Volume of Fluid (VOF) models to predict air entrainment, confirming their accuracy for hydroelectric intake applications. Meanwhile, Chen et al. (2017) [16] explored vortex helicity density and proposed mitigation strategies to suppress air entrainment through geometric modifications.

Recent studies have further refined numerical approaches to tackle specific challenges. Matsui et al. (2014) [17] investigated free surface instabilities in pump sumps through numerical simulations, identifying irregular flow patterns that caused air entrainment and reduced efficiency. Their findings suggested design modifications to control surface fluctuations and minimize free-surface vortices, enhancing sump stability. Similarly, Lei et al. (2019) [18] used the Lattice Boltzmann Method (LBM) with LES to classify vortex types, providing insights into optimizing sump dimensions to mitigate detrimental vortices and balance energy efficiency.

Amin et al. (2019) [19] explored the effects of variables such as inflow angle and sump depth on vortex formation, demonstrating that deeper sumps and wider designs reduced turbulence and improved stability. Yanase et al. (2019) [20] focused on air-entraining vortices, identifying critical submergence depths and flow velocities to minimize cavitation risks. Their recommendations for optimized submergence levels and inlet structures have practical implications for hydraulic efficiency.

Recent innovations have emphasized geometry optimization. Mohamed (2023) [21] examined submergence depth and sump width, showing that deeper configurations suppressed vortex formation and reduced air entrainment. Raj et al. (2024) [22] analyzed fluid flow using advanced CFD techniques, concluding that streamlined inlets and optimized sump depths significantly mitigated vortex formation and improved hydraulic stability.

Together, these experimental and numerical studies underscore the importance of integrating robust design principles with advanced computational techniques to optimize pump sump performance and minimize energy losses.

4. CFD Model

4.1. Model Description

The current research study employs Computational Fluid Dynamics (CFD) simulations to develop systematic guidelines and recommendations for best selection of the pump sump parameters: shape; arrangement; dimensions and other geometrical features to eliminate or minimize the previously described defects and drawbacks associated with the flow field.

ANSYS 2023 R1 is employed to solve the governing equations using the finite volume method. An implicit formulation is selected to convert the discretized equations to linear equations for the dependent variables in all computational cells in the domain. The suggested CFD model based on 3D two-phase numerical technique for the modelling of flow field in pump sump. For each phase, the model uses standard mass conservation principle, equations of momentum conservation and stress tensor considering the portion represents the volume dilation effect.

In the current numerical solution, the Realizable k-ε model is utilized to account for turbulence. Realizable k-ε Model closes turbulence and uses substitutional formula for solving turbulent viscosity. The equation of the transport in Realizable k-ε Model, modified for the effect of the rate of dissipation ε, was developed depending on transport equation for the fluctuation of the mean-square vorticity. Realizable k-ε Model is appropriate for flow modelling in pump sump due to its accuracy in simulating swirling and rapidly strained flows. When the primary-phase turbulence significantly influences the random motion of the secondary phases, the dispersed turbulence model becomes the model of choice.

4.2. Phases Interaction

The model in the current study solves liquid- gas domain. The Volume of Fluid (VOF) solver is selected to solve the flow motion for specified simulations. The VOF equations can simulate a minimum of two fluids on condition that they must be immiscible fluids. The results are obtained by resolving a single group of equations for momentum and also track the volume of the fraction for each fluid in the subjected domain.

VOF solver is suitable to be used for both steady state and time-dependent simulations. The results of the VOF solver for steady state simulations are reasonable if the solution is independent of the initial conditions and the initial flow conditions for the different phases are well defined. Therefore, two phase domain (air & water) with separate inlet boundaries of each phase can be solved using steady-state simulations using VOF solver.

The formulation of the VOF solver depends on the principle that every cell must contain two or more phases where these phases do not chemically interact with each other and are not immiscible.

For each phase that is added to the model, a new variable is added corresponding to this phase which represents the volume fraction of this phase to contribute in processing and filling the cells. For each cell, the sum of all phases fractions is equal to one. In each cell, the volume-averaged values of each variable are shared for all phases, while the volume fraction of all of the phases is detected in every single cell. The properties and variables in all cells are represented purely for one of the phases and also can be represented for the mixture of all phases in domain, based on volume fraction values in domain.

As example, if the α_q is the fluid's volume fraction in the cell of fluid noted q^{th}, then there must be one case of the following is valid:

$\alpha_q = 0$, in this case the cell is complete empty (i.e. doesn't have any volume of the q^{th} fluid).

$\alpha_q = 1$, The cell is completely full (filled with the q^{th} fluid).

$0 < \alpha_q < 1$, The cell contains a fraction of α_q out of 1 in the cell.

The equation of energy is shared among the phases, is shown below:

$$\frac{\partial}{\partial t}(\rho E) + \nabla.\left(\vec{v}(\rho E + p)\right) = \nabla.\left(k_{eff}\nabla T\right) + S_h \quad (1)$$

The VOF model treats energy, Temperature, as the traditional mass-averaged quantities:

$$E = \frac{\sum_{q=1}^{n} \alpha_q \rho_q E_q}{\sum_{q=1}^{n} \alpha_q \rho_q} \quad (2)$$

The method of Volume of Fluid (VOF) has some additional limitations and assumptions besides the main mathematical restrictions. These limitations can be summarized in the following points.

1. Solver of pressure-based approach has to be used since density-based solver is not valid for VOF.
2. In each cell in the entire domain, one or more phase of fluids must exist and no void region allowed.
3. If the domain includes gases, then one ideal gas phase must be used.
4. A periodic flow cannot be simulated while using the VOF solver.
5. Second order terms are not applicable with the VOF explicit buildup.
6. Both the DPM model and VOF model cannot run together.
7. Polyhedral meshes cannot be used in association with coupled VOF- Level-Set model.
8. VOF solver is suitable for premixed, non-premixed, partially premixed, and combustion models.

4.3. Flow Solver and Meshing

ANSYS uses one of two different solvers to solve the previously mentioned governing equations: pressure-based solver and density-based solver. The pressure-based solver is suitable for the low velocity incompressible flow field while density-based solver is suitable for high velocity compressible flow field. In pressure-based approach, the pressure is obtained by solving both momentum and continuity equations. The solver also solves (as required or applicable) energy equation and other non-vector variable governing equations like chemical species, suspended solids and or turbulence.

The computational domain is divided into volumetric cells by computational mesh (grid). The numerical approach includes a discretization process in which the finite volumes are generated.

Convergence is achieved if the residuals of flow parameters (flow rate, X velocity, Y velocity, Z velocity, turbulence, kinetic energy and volume fraction) reach below certain predefined values for all the concerned prementioned parameters. Iteration of the solution shall keep repeating until convergence is achieved.

The numerical simulations shall be performed with high resolution mesh to obtain results with reasonable quality and to obtain the fine details of the flow field in the pump sump specially the generated swirls and vortices.

The main parameter that controls model resolution is mesh size. The mesh size defines the maximum distance between nodes. To have high resolution, the mesh size should be as small as possible. Reducing the mesh size increases the simulation time and limit numerical computation capabilities and increase the requirements of the simulation processor. With decreasing mesh size, the accuracy of the numerical solution enhances until it reaches certain stable limits beyond which no more enhancement can be achieved with more decreasing of mesh size. Mesh with different sizes can present reasonable mesh size in the whole domain with fine mesh size near interesting areas in the domain. In most cases, large mesh size produces wrong results due to the iterative nature of the solution.

4.4. Numerical Model Calibration and Validation

The proposed numerical model in the current study is calibrated and validated by reproducing previously published laboratory experiments. The results obtained from the model are compared to the previous experimental results to test the ability of the model to simulate the flow pattern in pump sump with reasonable accuracy. Two previous experimental published research are used for the purpose of calibration and validation of the current CFD model; Arboleda and EL-Fadel (1996) and Okamura et al. (2007) [23], [24].

Arboleda and EL-Fadel (1996 [23]), have performed an experimental physical model for The

Natomas East Main Drain Canal (NEMDC) pump station which is implemented by Sacramento Area Flood Control Agency to develop flood control solution for drainage region in Sacramento. The experimental results are simulated by the developed numerical model using the same initial conditions. Quantitative experimental results are compared with the corresponding numerical results of the current study.

Figure (2) illustrates the geometry of the laboratory experiments to be simulated in this study; plan, sectional elevation, typical flow pattern for several pump operation scenarios and recirculation flow pattern. The figures are quoted from the original published research of Arboleda and EL-Fadel (1996) [4].

The experimental results of Arboleda and EL-Fadel (1996) [23] showed vortices at pumps' bell mouth locations. Some areas showed stagnation of water and other location showed an unsteady flow behaviour with flow moving in all directions.

To calibrate the proposed CFD model to simulate flow in pump sump, experimental results of Arboleda and EL-Fadel (1996) [23] have been compared with the reproduced CFD results for the same geometry.

Figure (3) shows the generated uniform distributed mesh all over the domain and the proposed boundary conditions.

Figure (4) shows the velocity distributions at horizontal section across the domain from CFD solution results. The velocity values from the CFD numerical results are in good agreement with the velocity values shown in the original experimental research paper. The figure also shows the vortices developed in sump. It is indicated in figure through a purple area and it is found at the last pump at the bottom as well as the non-working pump cells piers. The vortices in original research paper as shown in Figure (4) was found in same location at pipe but it is not showing vortices near the piers.

Figure (5) shows the distribution of flow velocity vectors from the CFD numerical simulation. The figure shows good agreement with the velocity vectors distribution of the published laboratory results shown in Figure (4). The figure shows high velocities at the entrance of domain, as well as stagnation on non-working pump bays which were also found in the previous experimental results.

Okamura, et al. (2007) [24] have performed laboratory experiment to reduce the drainage sumps volume and reducing the non-allowed vortices resulting in the sump due to this volume reduction. They performed laboratory experiments and the results were compared with CFD model. The positions of the vortices were obtained by the means of laser light sheets visualization. The vorticity and velocity distribution in sump has been measured by using a Particle Image Velocimetry (PIV) method and Laser Light Sheet (LLS) measuring device.

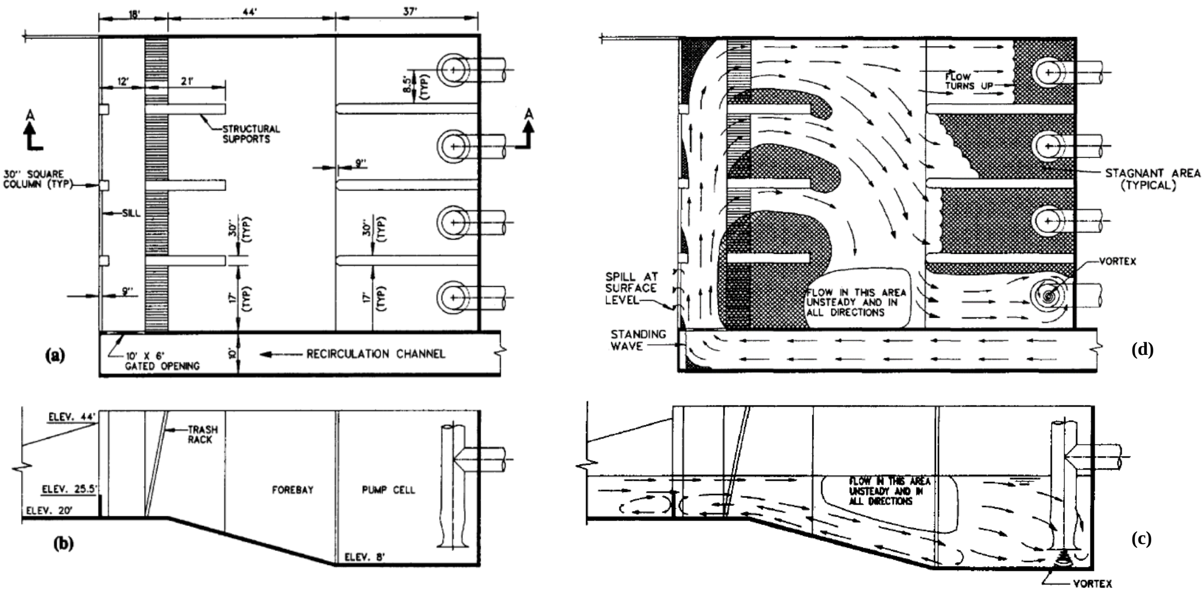

Fig.2: Sump and Approach Area Configuration: (a) Plan; (b) Sectional Elevation; (c) Typical Flow Pattern for One, Two, and Three Pump Operation; (d) Recirculation Flow Pattern at Floor Level (Plan View) (Source: Arboleda and EL-Fadel, 1996) [23]

In the current research, Okamura, et al. (2007) [24] experiment has been simulated numerically and the current numerical results show quite good agreement with their results for both laboratory experiments and their CFD simulated results. Figures 6 and 7 show the velocities and streamlines calculated through CFD codes and the measured laboratory results.

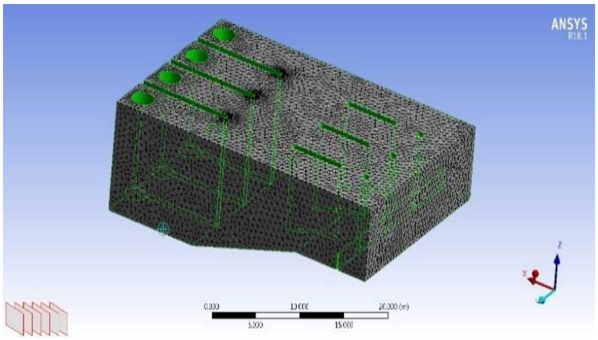

Fig.3: Meshing view of developed model

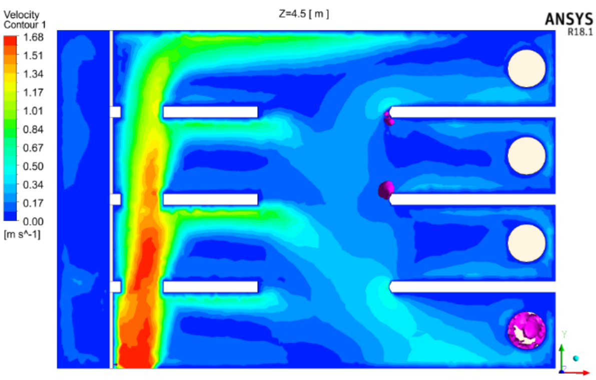

Fig.4: Flow Velocity Distribution contour with purple vortices.

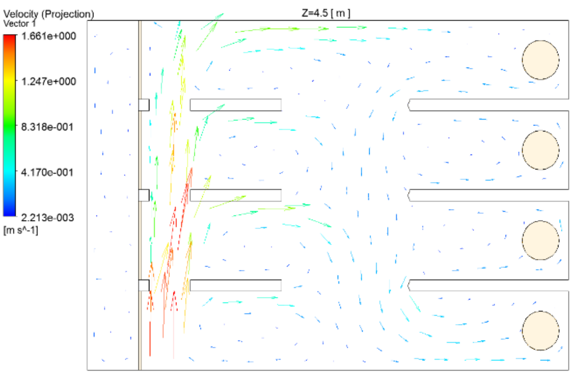

Fig. 5: Flow Velocity Vectors Distribution

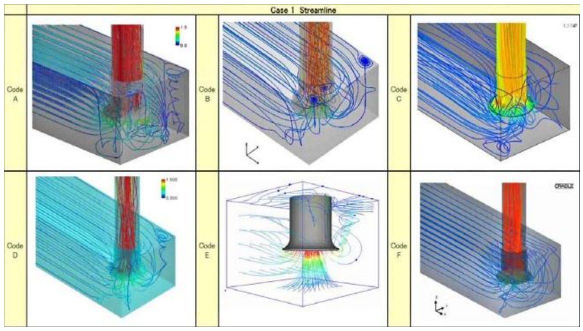

Fig.6: Comparison of streamlines calculated by different codes (Source: Okamura et al. 2007) [24]

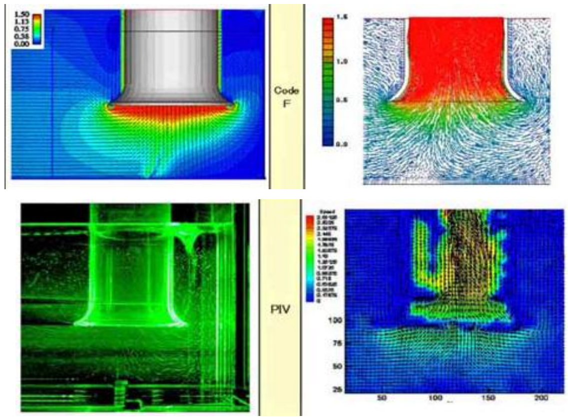

Fig.7: Calculated and measured velocities (Source: Okamura, et al. (2007) [24]

Figure 8 shows the uniform size mesh used in the current CFD model to reproduce Okamura, et al. (2007) [24] experiment. The streamlines obtained from the current CFD simulations (Figure 9) show excellent agreement with streamlines obtained from various CFD models experimented in Okamura, et al. (2007) [24]. Figure 10 shows the velocity vectors obtained from current simulations. Figure 11 shows the velocity distribution contours on section parallel to XZ plane as defined from the previous research paper used for verification. Comparing the Figures 10 and 11 with results available in original research paper in Figure 7, it is found that both velocity contours and vectors show excellent agreement with laboratory experimental results.

After simulating and comparing the experimental laboratory data published by Arboleda and EL-Fadel (1996) [23] and Okamura, et al. (2007) [24] with the results of the current developed CFD model it can be concluded that, the developed CFD model is capable of simulating the flow in pump sump with very good accuracy. Following calibration and validation of the developed model, systematic hypothetical simulations shall be performed to investigate the effect of the various parameters on the flow pattern in the pump sump and to select the most efficient geometry and other design parameter to eliminate the flow pattern defects that occur in the pump sump: vortices and swirls and water stagnation.

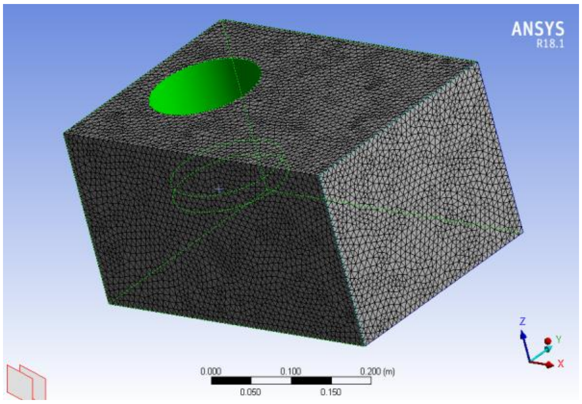

Fig.8: Meshing view of developed model

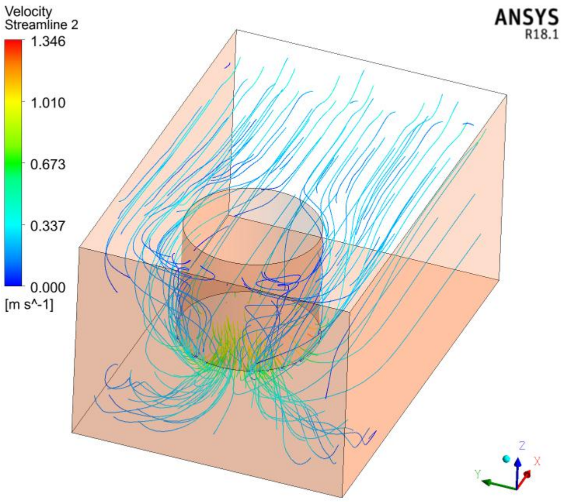

Figure 9: Calculated streamlines for developed model

4.5. Hypothetical Simulations

In the current study, the effect of the controlling parameters on the characteristics of the generated flow in pump sump is investigated through assumed hypothetical simulations solved using the developed numerical model. Various systematic values of input parameters and geometrical features and dimensions of the pump sump have been assumed and solved numerically using the developed model.

The most important affecting and controlling parameters of the flow pattern in pump sump have been selected based on previous experimental and numerical literature review and on practical experience. In the following table, the important controlling parameters of the flow pattern in pump sump are introduced. Each parameter listed in the table can have one or more values or selection as shown in the following table (1). In order to investigate the effect of each parameter separately, other parameters will be fixed during the simulations investigating that parameter.

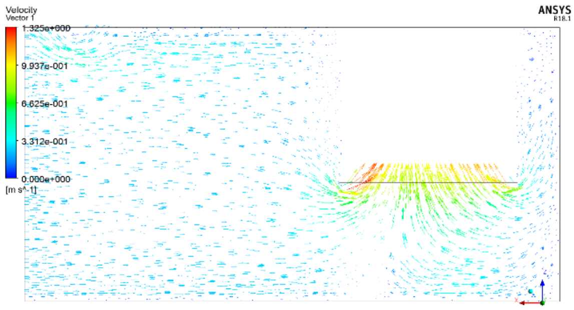

Figure 10: Calculated velocity vector for the developed model

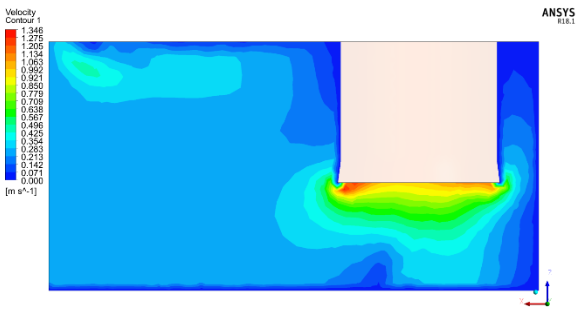

Figure 11: Calculated velocity contour for the developed model

Table (1): Selected Controlling parameters list.

	Parameter	Value(s)
1	Sump geometry	• Rectangular • Circular
2	Pump direction	• Axial • Horizontal
3	Inlet level	• Top • Bottom
4	Outlet level	• Top • Middle • Bottom

	Parameter	Value(s)
		- On the same level of inlet - Inverted from the inlet
5	Inlet/outlet position relation	• Same side • Opposite side • Perpendicular side
6	Number of operating Pumps	• 2 pumps • 3 pumps • 4 pumps - Can be on different sides
7	Number of Inlets	• 1 pipe • 2 pipes
8	Outlet Spacing	• Narrow (0.5m) • Wide (1.5m)

5. Analysis Of Flow Pattern Dominating Parameters: CFD Hypothetical Simulations

The three problems that are associated with the flow pattern in pump sump as described before are: vortices and swirl generation, stagnation, unequal suction of pumping units. In the following sections, the effect of geometrical features, dimensions and operational conditions on the generated problems of the flow pattern in pump sump shall be investigated through hypothetical numerical simulations. This is achieved by changing only one parameter and fixing all other parameters through the performed simulations.

5.1. Effect on Vortices and Swirls Formation

Formation of vortices in pump sump reduces pumps efficiency and may cause vibrations and instability of pumps operation. This effect reduces the pumps lifetime and increases the need for cyclic maintenance. The vortices are usually found near suction pipes, inlet pipes, and sidewalls. The effect of each above-mentioned parameters on the formation and properties of the formed vortices and swirls are presented herein. The results of the numerical simulations are represented by rendering the vortex core in a 3D sump at constant swirling strength.

5.1.1. Sump Geometry

Figures 12 and 13 show the vortex formed in circular sump and rectangular sump with 3 vertical suction pipes representing vertical turbine pumping units and one inlet pipe at the bottom of the sum. For circular sump, figure 12 shows small vortices formed at suction pipes flow entrance, as well as low or no vortices at inlet pipe entrance. For rectangular sump, figure 13 shows vortices formed at the suction pipes entrance as well as the inlet pipe sump entrance. The two-phase flow modeling allows simulating air entrained vortices from air into water. The previous figures show that changing the sump's geometry from rectangular to circular shape reduces the vortex intensity and counts as the vortex at the middle pump and the vortex at the distance between the inlet pipes and the middle pump have vanished.

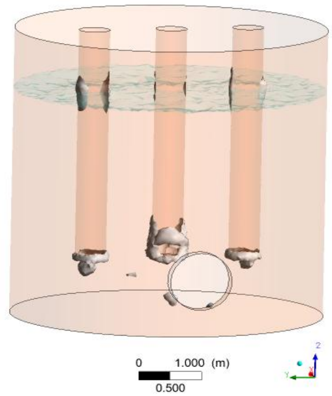

Fig. 12: Vortex core in circular sump

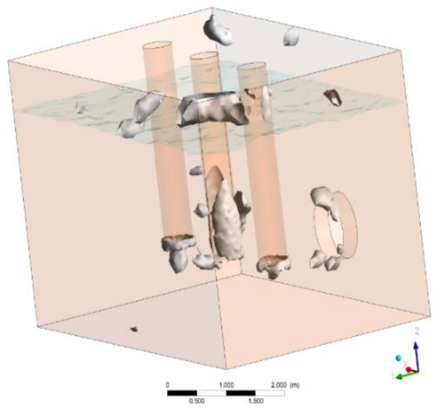

Fig. 13: Vortex core in rectangular sump

5.1.2. Pump Suction Direction

Vortex formation is shown in Figures 14 to 17 for different pump suction directions, vertical and horizontal, represented by axial vertical pumps and centrifugal horizontal pumps for both considered shapes of pump sump: circular and rectangular. In figures 14 and 15 for circular pump sump, less and weaker vortex formation is found for horizontal suction direction. In figures 16 and 17 for rectangular pump sump, vortices are formed at vertical suction direction entrance while no vortices formed in the sump for horizontal suction direction. The vortices shown in figure 17 are some false rounded vortex cores at suction pipes entrances. The reduction in water level for horizontal suction also indicates that horizontal pumps have relative better suction rate than vertical pumps. The four previous figures show that in typical sumps with different pump suction direction, the use of horizontal pumps instead of vertical pumps reduce the vortices formation at all locations inside sump. In addition, as mentioned above, using circular sump reduce the vortices intensity and eventually vortices disappear in sump.

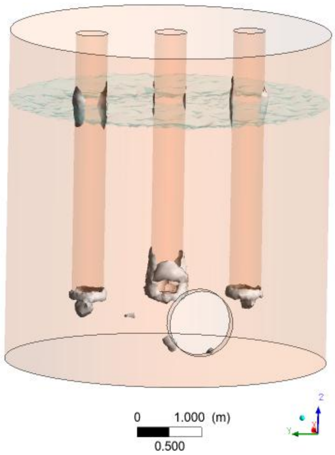

Fig.14: Vortex core at vertical pumps (in circular sump)

Fig. 15: Vortex core at centrifugal pumps (in circular sump)

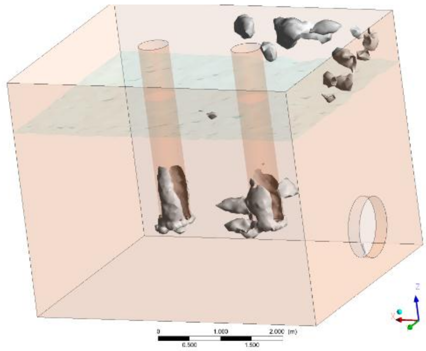

Fig.16: Vortex core at vertical pumps (in rectangular sump)

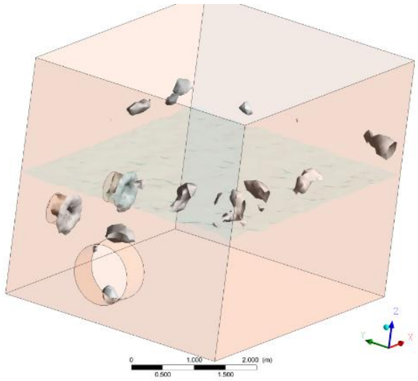

Fig.17: Vortex core at centrifugal pumps (in rectangular sump)

5.1.3. Inlet Level

Improper level of the inflow to the pump sump may lead to creation of massive vortex in sump and or even make the entire water body rotates inside the sump. The vortices formed in rectangular pump sump for low-level inlet and for high-level inlet are shown in Figures 16 and 18. Figure 16 shows vortices formed at suction entrance as well as beside the first suction pipe. The side vortex is typically denoted to be formed at suction pipe entrance and pushed away by the main stream coming from the inlet pipe. This means that even larger vortices developed at suction pipe entrance. Figure 18 shows no vortices formed at first suction pipe and relatively small vortex at the second suction pipe with respect to the vortices formed in sump with low-level inlet.

There may be one more situation where the inlet pipe is normal to pump alignment. Therefore, the next set of figures compare the vortices developed in previous two sumps, but with inlet pipes normal to pump alignment. Sump shown in Figure 19 is the same as shown in Figure 16 except for the inlet direction. The sump in Figure 19 has an inlet pipe directed normal to pumps alignment. This distribution gives the flow the ability to be evenly distributed through suction pipes. The vortices are noticed to be considerably smaller than the vortices formed at sump with inlet pipe aligned with pump.

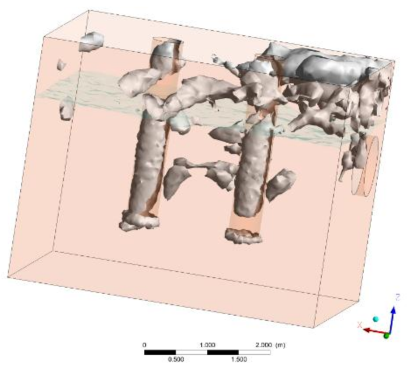

Fig.18: Vortex core in sump with high inlet level.

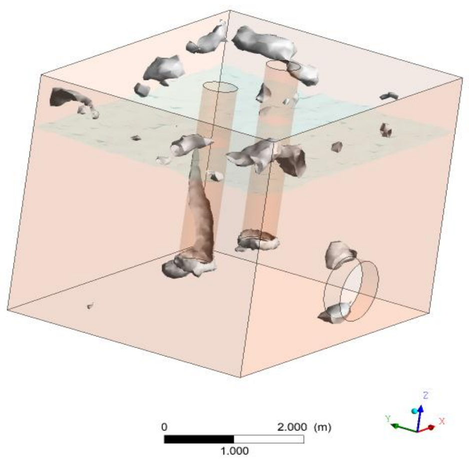

Fig.19: Vortex core in sump with low inlet level (normal direction)

Sump shown in Figure 20 is the same as shown in Figure 18 except for the inlet direction. The sump in Figure 20 has an inlet pipe directed normal to pumps alignment. The vortices formed in both sumps are almost typical at suction pipes entrance. Both cases in Figures 18 and 20 show fewer vortices development at suction pipes than the other two cases in Figures 16 and 19.

Higher inlet level reduces the vortices developed in pump sump. Planning sump geometry with inlet pipe direction normal to the suction pipes axis direction slightly reduces the developed vortices.

5.1.4. Inlet/Outlet Position Relation

When planning a pump sump layout, inlet/outlet position must be placed to achieve stable flow pattern in the sump. The Inlet/outlets position can be either on same side, opposite sides or at different sides of sump. This parameter can only apply while using horizontal suction pipes. In the following part, the effect of inlet/outlet position on vortices formation is investigated.

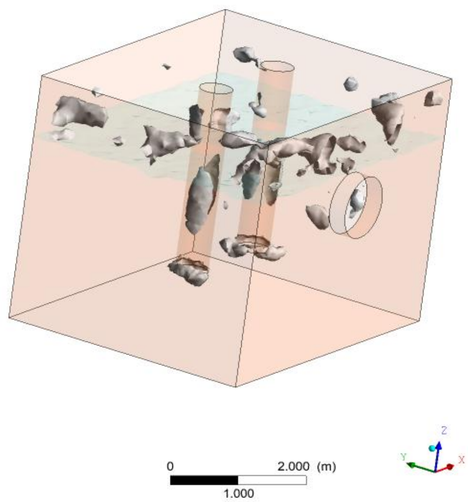

Fig. 20: Vortex core in sump with high inlet level (normal direction)

Figure 21 shows rectangular sump with horizontal suction and inlet pipes on same side and level. Vortices is developed at suction pipe entrance and at inlet pipe entrance in addition to rotating mass on the far side from the inlet and outlet side. Figure 22 shows a rectangular sump with suction and inlet pipes and on the opposite side the inlet pipe exists. The figure shows no vortices developed in the sump but some false vortex cores at suction pipes entrance. Setting the suction pipes on the opposite side with inlet pipes considerably decreases formation of vortices and decreases their intensity around the suction pipes and on other locations in sump.

5.1.5. Number of Pumping Units

Figures 23, 24 and 25 show the vortices formed in rectangular sump with two, three, and four vertical suction lines representing vertical pumps. Vortices are developed at suction pipes, and they considerably decrease in size and intensity for multiple suction pipes. Increasing number of pumps reduces the vortices intensity, especially at the pump far from the inlet pipe.

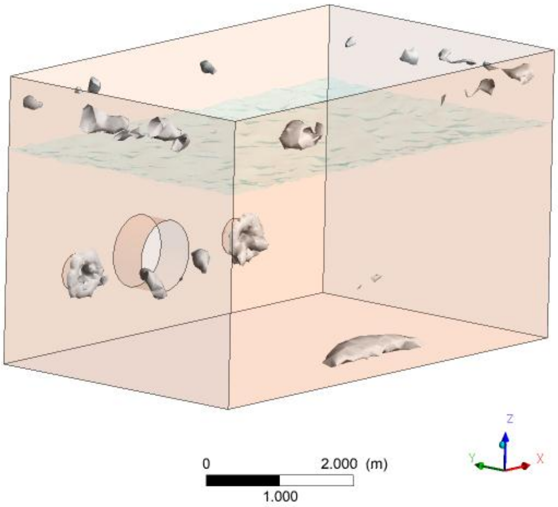

Fig.21: Vortex core in sump with inlet on same side of outlets

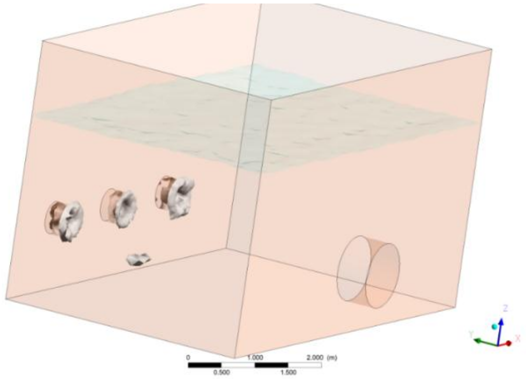

Fig.22: Vortex core in sump with inlet on opposite side of outlets.

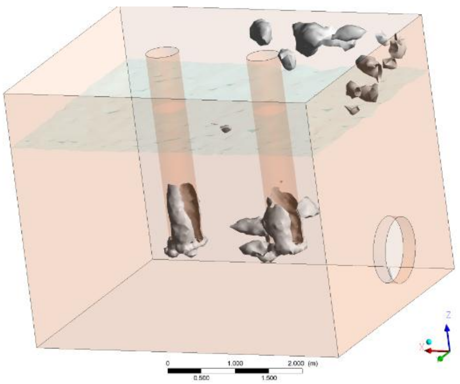

Fig. 23: Vortex core in sump with two axial pumps

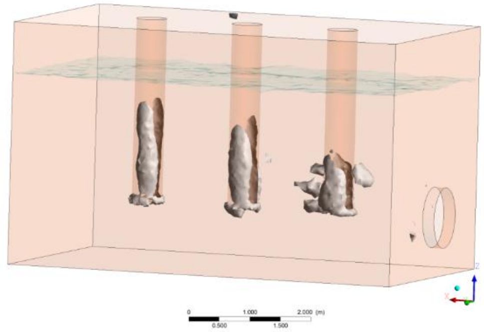

Fig. 24: Vortex core in sump with three axial pumps

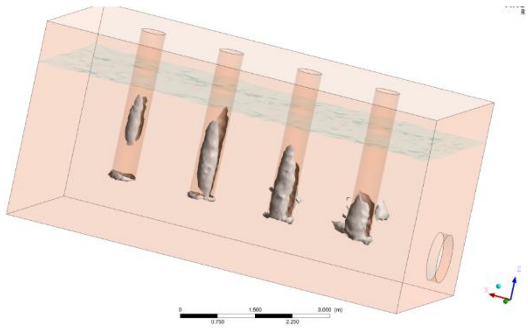

Fig. 25: Vortex core in sump with four axial pumps

5.1.6. Number of Inlet Pipes

Proposing more than one inlet to the pump sump can have considerable effect on vortices formation. Figures 26 and 27 show rectangular sump with two vertical suction pipes and two and one inlet pipes. The sump has vortices developed at suction pipe entrance as well as pipe inlet. The count of vortices formed in sump with two inlets are greater than those formed in sump with one inlet, however, the vortices size and intensity are less. Having more than one inlet to the pump sump increases the vortices count but with lower intensity. In addition, if inlets are poorly arranged, this may lead to total mass rotation in sump.

5.1.7. Suction Outlet Level (Submergence)

Pump submergence is one of the most important parameters that affects sump vortices formation. Three outlet levels (submergence) shall be experimented herein to investigate its effect on vortices formation. Figures 28, 29 and 30 show the vortices formed in rectangular sump with different submergence. Vortices are formed at suction pipes entrance, as well as intensive side vortices formed all over the length of the sump facing inlet pipe. Increasing level of outlet suction (i.e. decreasing submergence) decreases the vortices formed along the suction pipes and decreases the side vortices. For more increase of outlet suction level above the inlet level, minor vortices are formed along suction pipes and no side vortices are formed. The most intense vortices are formed if the inlet and outlet are in the same levels.

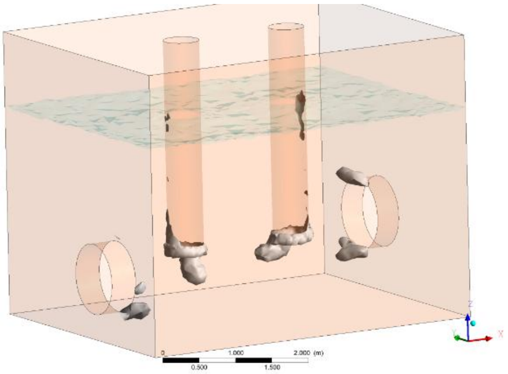

Fig.26: Vortex core in sump with two inlet

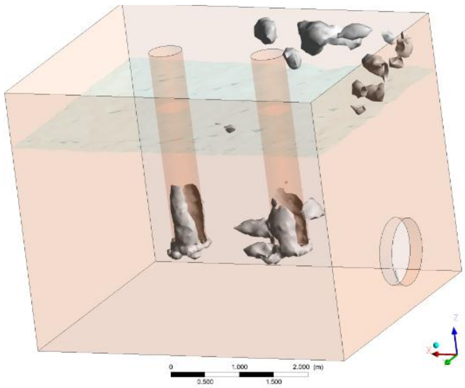

Fig.27: Vortex core in sump with one inlet pipes

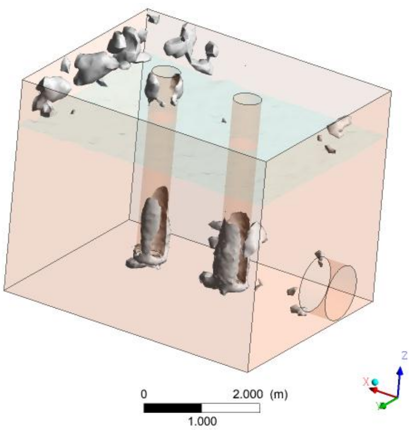

Fig. 28: Vortex core in sump with low level outlet pipes (below inlet level)

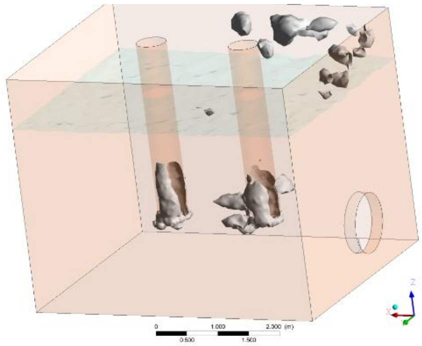

Fig.29: Vortex core in sump with medium level outlet pipes (same inlet level)

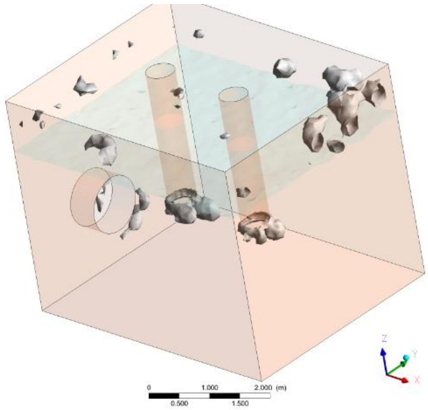

Fig. 30: Vortex core in sump with high-level outlet pipes (above inlet level)

5.1.8. Outlets Spacing

The spacing of the outlets suction pipes is one of the main features of sump layout considerations and it has a major effect on flow pattern and vortices formation in pump sump. Figures 31 to 32 show vortices formed in rectangular sump with different spacing for outlet suction pipes. Figures 33 to 34 show vortices formed in circular sump with different spacing for outlet pipes. For small spacing between outlet pipes, large and intense vortices are formed between suction pipes beside side vortices formed around pipes entrance. Increasing the spacing between outlet suction pipes decreases the formed vortices in size and in intensity and therefore, the sump has a better suction performance.

5.2. Effect on Stagnation Zones

Stagnation in pump sump is a major problem especially for freshwater sumps. The existence of stagnant water zone in sump reduces the pumped water quality, reduces the mixing of water and increase the need for periodic cleanup for sump as it suffers more sedimentation. In the following analysis,

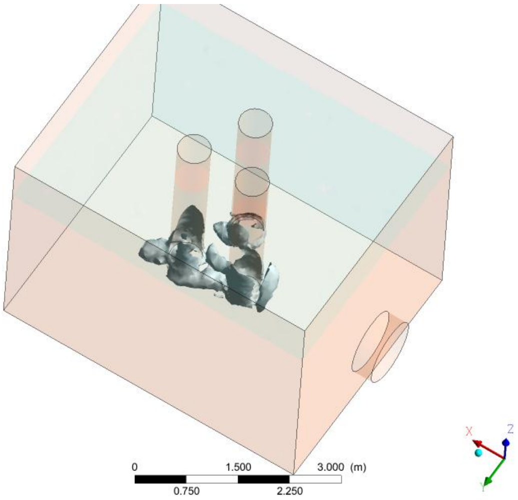

Fig.31: Vortex core in sump with low spacing between outlets

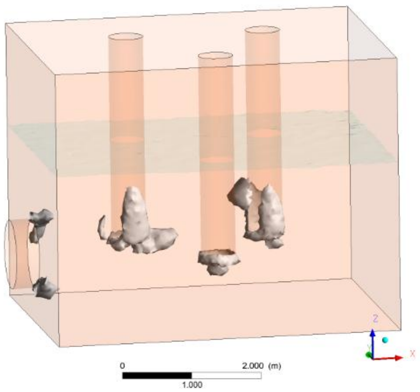

Fig. 32: Vortex core in sump with high spacing between outlets

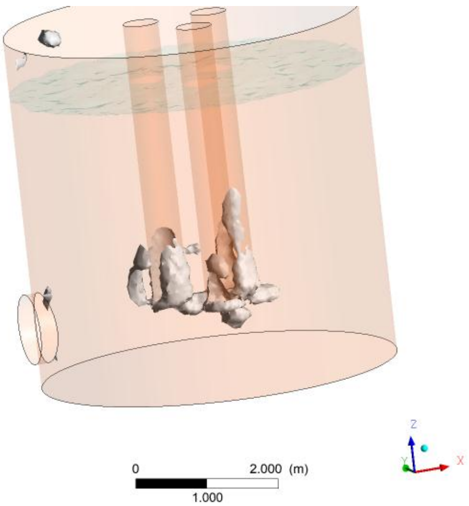

Figure 33 Vortex core in sump with low spacing between outlets (circular sump)

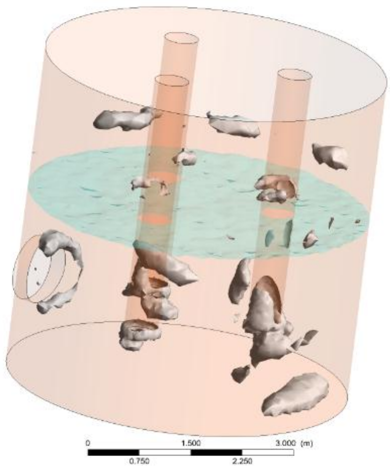

Fig. 34: Vortex core in sump with high spacing between outlets (circular sump).

stagnation is considered if the magnitude of total flow velocity is less than 0.1 m/s. The water bodies are identified to be stagnant if they have different colour than blue in sump. Also, water in sump has a transparency effect so that stagnant bodies can be shown even inside the water volume.

5.2.1. Sump Geometry

Pump sump Geometry should be selected to prevent or minimize stagnation zones within the sump. However, in most cases sump geometry is planned according to construction aspects. Figures 35 and 36 shows circular sump and rectangular sump with vertical pumps and bottom inlet directed perpendicular to the suction pipes alignment line. The figures show that stagnation is minimal inside circular sump while stagnation zones occur in most of the areas in rectangular sump where flow moves directly from the inlet pipe towards the vertical suction pipes.

5.2.2. Pump Alignment Direction

Figures 37 and 38 shows rectangular sumps with vertical suction pumps and with horizontal suction pumps respectively with inlet pipe beside the bottom of the sump. In Figure 37, stagnation occurs in most of the areas in case of vertical suction pumps. Sump with horizontal suction pipes exhibit considerably less stagnation as shown in Figure 38. Using horizontal suction pipes instead of vertical suction pipes significantly reduces the stagnant water zone existence in the sump even if for rectangular sumps. Stagnation is even reduced in case of circular sumps.

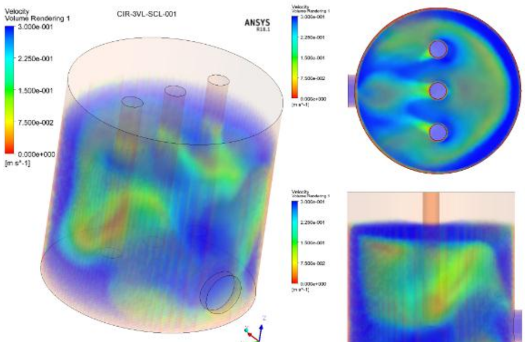

Fig. 35: Stagnant water bodies in circular sump

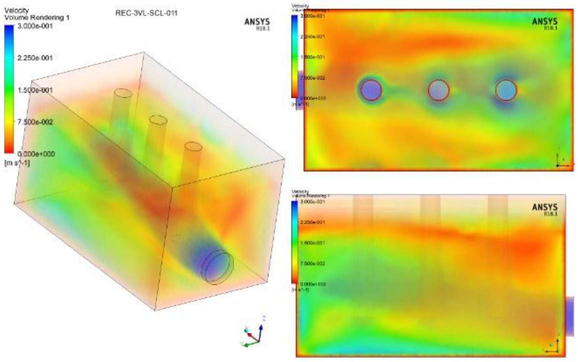

Fig. 36: Stagnant water bodies in rectangular sump

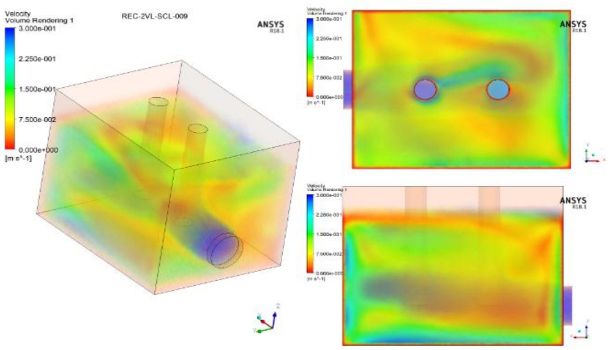

Fig.37: Stagnant water bodies in sump with axial pumps

5.2.3. Inlet Water Level

Inlet water level is one of factors can eliminate or reduce stagnant water bodies in pump sump. Figures 37 and 39 show rectangular sumps with low inlet level at the same level as the vertical suction pipes and high inlet water level above the level of the vertical suction pipes. Stagnation occurs in most of the areas in the sump with low inlet water level since water flows directly from inlet pipe to suction outlet pipes directly since they are on the same level. Sump with high inlet water level exhibit considerably less stagnation (Figure 39), however, there are still stagnant areas surrounding the line connects the inlet pipe and suction pipes. Forming the inlet pipe to be perpendicular to the pump alignment line shall significantly reduce the stagnation zones for both low inlet level and high inlet level (Figures 40 and 41). Some noticeable stagnant bodies located near the sump corners and below the inlet pipe (at the bed) can still be noticeable.

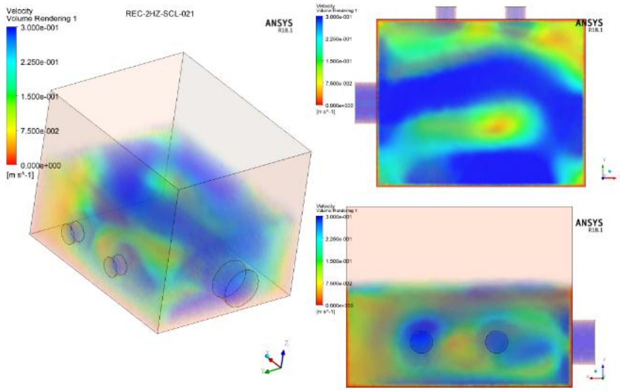

Fig. 38: Stagnant water bodies in sump with centrifugal pumps

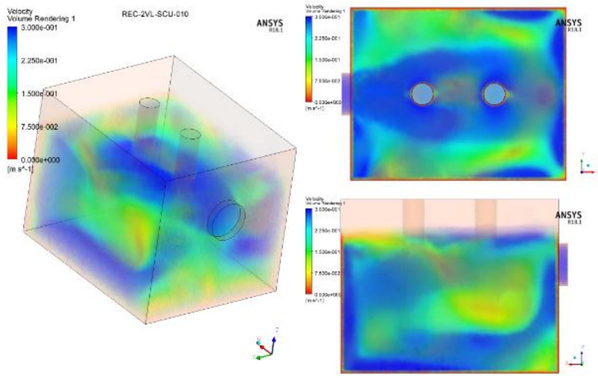

Fig.39: Stagnant water bodies in sump with axial pumps (high-level inlet)

Figures 42 and 43 show rectangular sumps with low inlet level at the same level as the vertical suction pipes and high inlet water level above the level of the horizontal suction pipes. Placing the inlet pipe level at different level from the suction pipes level, considerably less stagnation occurs within the sump. It is a good practice to have the inlet pipe at high elevation and far from the suction pipes.

However, for rectangular sumps stagnant water zones are noticed at the corners due to the sharp corners of the sump. Circular sumps do not have sharp corners and therefore, no stagnation is developed.

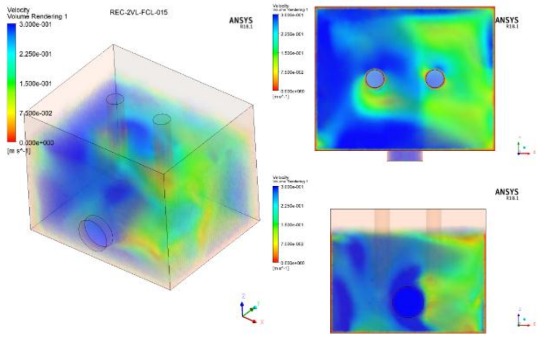

Figure 40 Stagnant water bodies in sump with axial pumps (normal inlet direction)

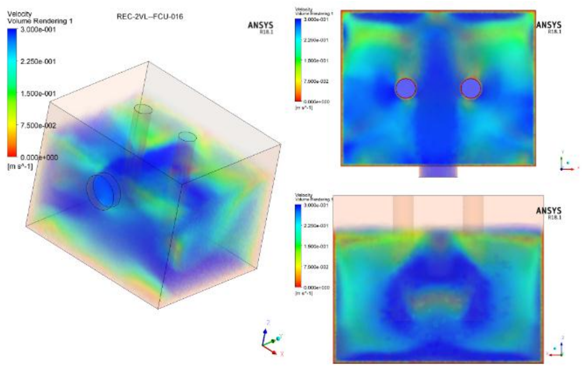

Figure 41 Stagnant water bodies in sump with axial pumps (normal, high-level inlet)

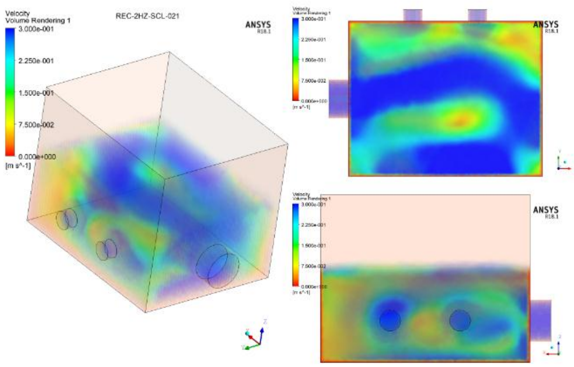

Figure 42 Stagnant water bodies in sump with centrifugal pumps (low-level inlet)

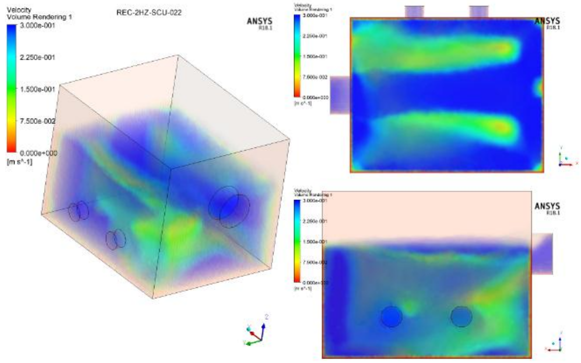

Figure 43 Stagnant water bodies in sump with centrifugal pumps (high-level inlet)

5.2.4. Inlet/Outlet Position Relation

The effect of the inlet/outlet position on forming stagnation and dead zones within the sump is investigated herein. The inlet pipe direction can be normal or parallel to the outlet suction pipes axis. The effect of existence of suction pipes and inlet pipes on same side or on opposite side is also investigated.

Figures 44 and 45 show rectangular sump with three suction horizontal pipes with inlet pipe at the opposite side and on the same side of the suction pipes respectively (normal to pipe alignment). Significant stagnation all over the sump occurs if the suction and inlet pipes are at opposite sides while stagnation considerably reduced at the corners if suction and inlet pipes are at same side since the flow is forced to circulate through the sump until it reaches the outlet pipes.

Figures 37 and 40 show rectangular sumps with vertical suction pipes and inlet pipe beside the bottom of the sump. In Figure 37 where the inlet pipe direction is parallel to suction pipes axis, the sump shows poor flow circulation, and the only non-stagnant part is the path between the inlet pipe and the suction pipes. In Figure 40 where the inlet pipe axis is perpendicular to the suction pipes axis, stagnation significantly reduced with some noticeable stagnant bodies located near the sump corners and below the inlet pipe at the bed.

Concluding the results shown in previous figures, it is found that using inlet pipes directed normal to suction pipes axis as well as on the same side with them reduces the existence of stagnant water bodies in sumps.

5.2.5. Number of Working Pumps

Number of working pumps is one of important variables that may be adopted to enhance the sump performance and reduces stagnation zones since it varies the exit points for the water from sump. Figures 36 and 37 show rectangular sumps with vertical suction pipes and inlet pipe beside the bottom of the sump; three vertical suction pipes in Figure 36 and two vertical pumps in Figure 37. Both figures show the only non-stagnant part is the water body located between the inlet pipe and the suction pipes.
Same results are shown in Figure 46 for four suction pipes. This indicates that changing the number of pumps has no significant effect on stagnant water bodies formation in pump sumps.

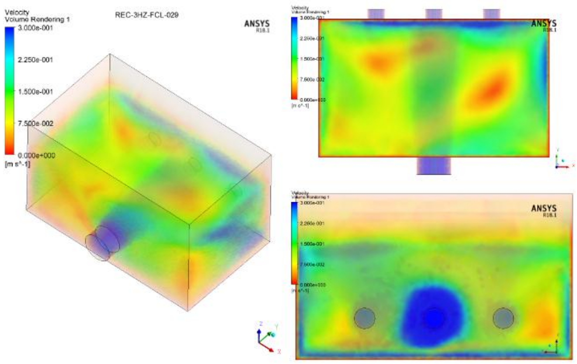

Fig. 44: Stagnant water bodies in sump (inlet on opposite side of outlets, low level)

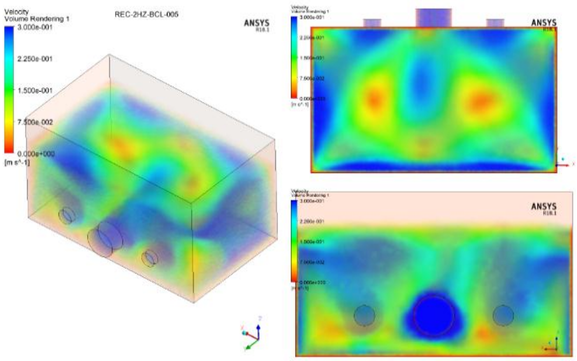

Fig.45: Stagnant water bodies in sump (inlet on same side of outlets, low level)

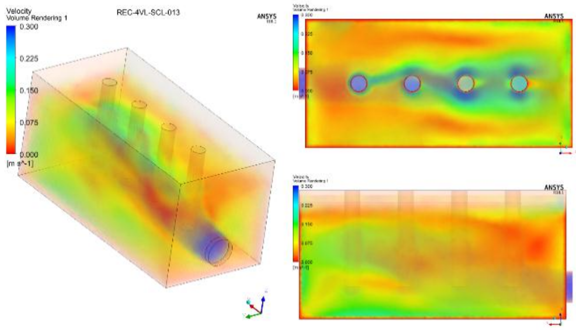

Figure 46 Stagnant water bodies in sump (four axial pumps)

5.2.6. Number of Inlet Pipes

Number of inlet pipes can be manipulated to enhance the flow pattern in pump sump. Figures 37 and 47 shows a rectangular sump with two vertical suction pipes with one inlet pipe and two inlet pipes respectively. The inlet pipes are set at the bottom. The two inlet pipes shown in Figure 47 are set in opposite direction and not facing each other. Poor circulation and considerable stagnation zones are developed in Figure 37 with one inlet pipe.

Figure 47 shows considerable enhancement in flow circulation in sump with two inlet pipes. The only locations that are still developing stagnant bodies are the corners and the area between suction pipes which are relatively small relating to the volume of water in sump. Stagnation zones can be considerably reduced or eliminated by using multiple inlets in sumps.

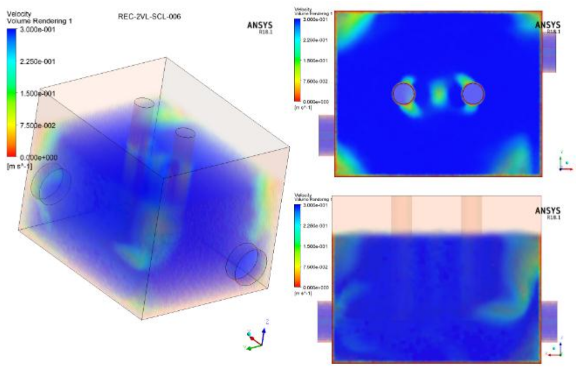

Fig.47: Stagnant water bodies in sump (two inlet pipes)

5.2.7. Suction Outlet Level (Submergence)

To investigate the effect of the outlet level on stagnation, three different outlet levels are defined: low level, medium level, and the high level (0.5, 1.0 and 1.5 m above the bed respectively). Figures 37, 48 and 49 show rectangular sump with two vertical suction pipes with different outlet levels. For low level suction pipes, the sump is almost fully stagnant, except for the area that connects the inlet pipe and the suction pipes. For medium level suction pipes, less stagnation occurs in the sump and minor enhancement in the flow pattern is achieved. For high level suction pipes, more flow movement occurs in the sump since the change in outlet level enforce more flow circulation inside the sump.

Suction Outlet Level is not flexible factor for manipulation since it usually controlled by other aspects in site such as water level in the source and submergence requirements. The results show minor effect of suction level on reducing stagnation in the sump. Suction outlet level parameter is found to have minor effect on enhancing flow pattern and reducing stagnation within the pump sump.

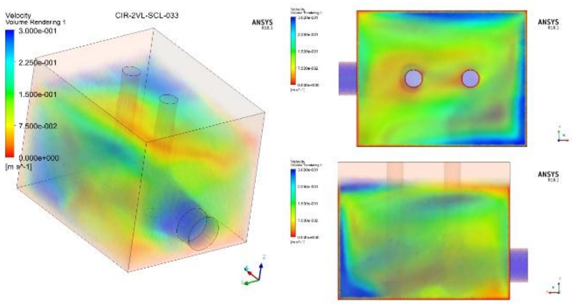

Figure 48 Stagnant water bodies in sump (medium outlets level)

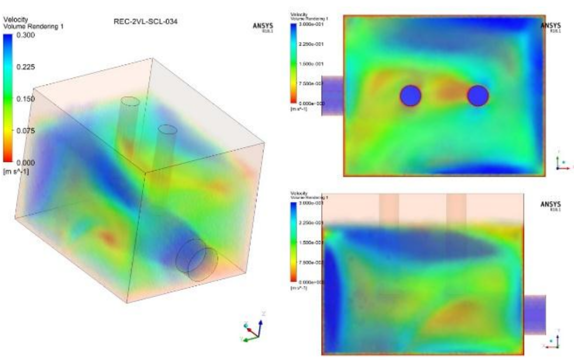

Fig.49: Stagnant water bodies in sump (high outlets level)

5.2.8. Outlet Spacing

Outlet spacing is considered an important parameter that affect stagnation in pump sump. Two configurations for outlet spacing are assumed herein; small spacing (0.5m) and large spacing (1.5m). It also worth mention that the rest of the study used (1.0m) as the standard spacing. Figures 50 and 51 show circular sump that has three suction pipes with large spacing and small spacing respectively. Less stagnation exists with large spacing however, both sumps suffer from stagnation except part of the flow between the inlet pipe and the outlet pipes as well as a thin area located on the opposite side from the inlet pipe (near the sump wall). Figures 52 and 53 show rectangular sump that has three suction pipes with large spacing and small spacing respectively. Rectangular sumps show similar behaviour to circular sumps. Using a large outlet spacing results in better water movement in sump, better flow pattern, enhanced suction performance and less stagnant water bodies.

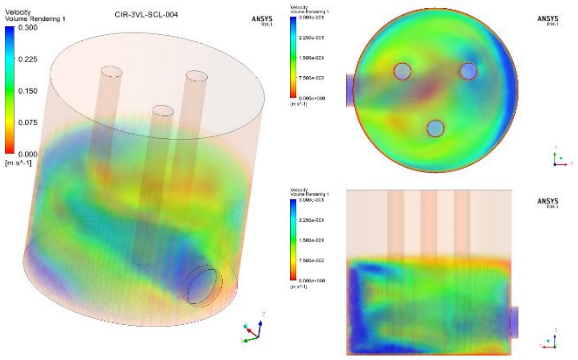

Fig.50: Stagnant water bodies in sump (large outlet spacing, circular. Sump)

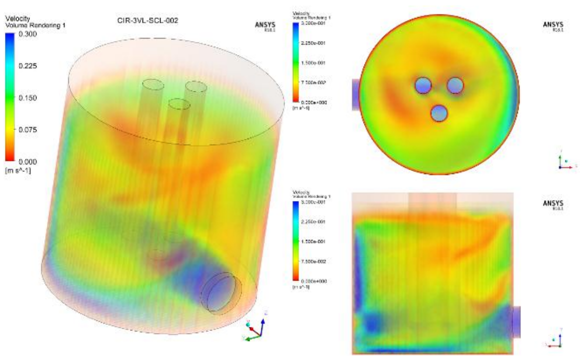

Fig.51: Stagnant water bodies in sump (small outlet spacing, circular sump)

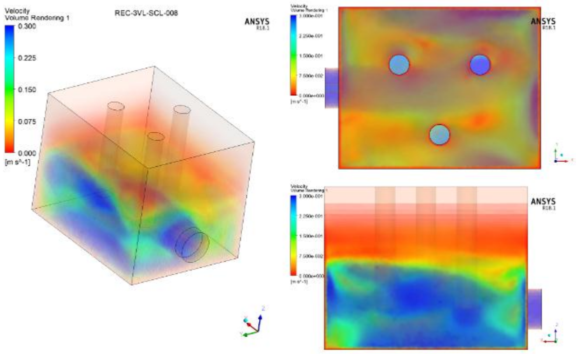

Fig.52: Stagnant water bodies in sump (large outlet spacing, rectangular sump)

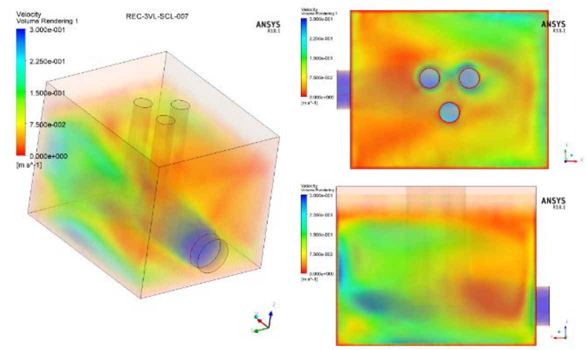

Figure 53 Stagnant water bodies in sump (small outlet spacing, rectangular. Sump)

6. Conclusion and Recommendations

The current study defined the major problems that occur in the flow field in pump sump: vortices and swirls and stagnation. The current study defined the major controlling factors that may affect the flow problems in pump sump: sump geometry, pump suction direction, inlet level, inlet/outlet position relation, number of operating pumping units, number of inlet pipes, suction outlet level (submergence) and outlet spacing. Using numerical solution and CFD simulations, the effect of each controlling parameter on the flow field pattern and formation of vortices and swirls and stagnation is investigated using several hypothetical sump geometry and flow cases. Each parameter is changed with two or more values individually in separate simulations to investigate its explicit effect on the flow field and the consequent vortices and swirls and stagnation.

Circular sumps reduce the developed vortices inside the sump especially the vortices formed in front of inlet pipes, unlike the rectangular sumps. Circular sumps increase water circulation in sumps and reduces the stagnant water zones unlike the rectangular shaped sumps.

Pump suction direction is a major parameter affecting developing vortices in sump. Using horizontal suction reduces the vortices formation while the vertical suction allows considerable vortices formation. Horizontal suction forces the flow through the sump to reach the suction points which reduces stagnant water zone while vertical suction increases stagnant water zones.

High-level inlet pipes is found to reduce the developed vortices especially the vortices formed in front of suction pipes. Low-level inlet pipes have shown higher vortices formation near the suction pipes. Inlet pipe directed normal to suction pipes alignment develop less vortices than those formed if inlet pipe is parallel to the suction pipes. Using high-level inlets reduces stagnant water zones within the sump.

Less vortices are formed in case inlet pipe and outlet suction pipes are on opposite sides of the sump while more vortices are formed when inlet pipe and the outlet suction pipes are on the same side of the sump. Stagnation zones are less in case inlet pipe and the outlet suction pipes are on the same side.

Less vortices are formed within the sump with increasing the number of suction pipes (number of working pumps). The number of suction pipes has no significant effect on stagnation zones within the sump.

Having several inlet pipes on opposite sides reduces the vortices intensity developed in sump but increases their count. Increasing number of inlet pipes to the sump significantly reduces the stagnation zones.

Intense vortices are formed within the sump, if inlet and suction pipes are on the same level. Varying the levels of inlet and suction pipes reduces vortices in count and intensity and also reducing stagnation zones formed within the sump.

Increasing outlet pipes spacing (or distance between pumps axes) reduces the intensity of the developed vortices especially those around the suction pipes entrance. Increasing outlet pipes spacing reduces water stagnation and enhances the flow pattern in pump sump.

Further numerical CFD Investigations can be performed on vortices formation and stagnation with anti-vortex countermeasures, separator wall inside the sump. Also, other controlling factors can be introduced such as inlet/outlet diameter ratio its effects on vortices formation and stagnation.

Acknowledgements

I would like to express my gratitude to my advisors, the participants, colleagues, and family for their support and contributions to the research project. Thank you all for your invaluable assistance and encouragement.

References

[1] Padmanabhan, M. & Hecker, G. E., "Scale Effects in Pump Sump Models," Journal of Hydraulic Engineering, vol. 110, no. 11, pp. 1540–1556,1984, doi: 10.1061/(ASCE)0733-9429(1984)110:11(1540).

[2] Odgaard, A. J. & Dlubac, J. J., "Hydraulic Model Study of Pump Sump Design," Journal of Hydraulic Engineering, vol. 110, no. 9, pp. 1267–1272,1984, doi: 10.1061/(ASCE)0733-9429(1984)110:9(1267).

[3] Hite, J. E. & Mih, W. C., "Velocity of Air- Core Vortices at Hydraulic Intakes," Journal of Hydraulic Engineering, vol. 120, no. 3, pp. 284–297,1994, doi: 10.1061/(ASCE)0733-9429(1994)120:3(284).

[4] Arboleda, G. & El-Fadel, M., "Effects of Approach Flow Conditions on Pump Sump Design," Journal of Hydraulic Engineering, vol. 122, no. 9, pp. 489–494,1996, doi: 10.1061/(ASCE)0733-9429(1996)122:9(489).

[5] De Fockert, A., Van 't Westende, J. M. C., & Verhaart, F. I. H., "Automatic swirl angle measurements for pump intake design," Journal of Hydraulic Research, vol. 53, no. 3, pp. 384–393,2015, doi: 10.1080/00221686.2014.989459.

[6] Ansar, M. & Nakato, T., "Experimental Study of 3D Pump-Intake Flows with and without Cross Flow," Journal of Hydraulic Engineering, vol. 127, no. 10, pp. 825–834,2001, doi: 10.1061/(ASCE)0733-9429(2001)127:10(825).

[7] Sarkardeh, H., Zarrati, A. R., & Roshan, R., "Effect of intake head wall and trash rack on vortices," Journal of Hydraulic Research, vol. 48, no. 1 EXTRA ISSUE, pp. 108–112,2010, doi: 10.1080/00221680903565952.

[8] Taştan, K. & Yildirim, N., "Effects of Froude, Reynolds, and Weber numbers on an air-entraining vortex," Journal of Hydraulic Research, vol. 52, no. 3, pp. 421–425,2014, doi: 10.1080/00221686.2013.879541.

[9] Suerich-Gulick, F., Gaskin, S. J., Villeneuve, M., & Parkinson, É., "Free surface intake vortices: Scale effects due to surface tension and viscosity," Journal of Hydraulic Research, vol. 52, no. 4, pp. 513–522,2014, doi: 10.1080/00221686.2014.905503.

[10] Sun, H. & Liu, Y., "Theoretical and experimental study on the vortex at hydraulic intakes," Journal of Hydraulic Research, vol. 53, no. 6, pp. 787–796,2015, doi: 10.1080/00221686.2015.1076533.

[11] Shinde, P. & Arakerimath, R. R., "Numerical approach for hydraulic performance formulation of centrifugal pump sump," 2023, p. 020073. doi: 10.1063/5.0100877.

[12] Neary, V. S., Sotiropoulos, F., & Odgaard, A. J., "Three-Dimensional Numerical Model of Lateral-Intake Inflows," Journal of Hydraulic Engineering, vol. 125, no. 2, pp. 126–140,1999, doi: 10.1061/(ASCE)0733-9429(1999)125:2(126).

[13] Ge, L. & Sotiropoulos, F., "3D Unsteady RANS Modeling of Complex Hydraulic Engineering Flows. I: Numerical Model," Journal of Hydraulic Engineering, vol. 131, no. 9, pp. 800–808,2005, doi: 10.1061/(ASCE)0733-9429(2005)131:9(800).

[14] Tokyay, T. E. & Constantinescu, S. G., "Validation of a Large-Eddy Simulation Model to Simulate Flow in Pump Intakes of Realistic Geometry," Journal of Hydraulic Engineering, vol. 132, no. 12, pp. 1303–1315,2006, doi: 10.1061/(ASCE)0733-9429(2006)132:12(1303).

[15] Nakayama, A. & Hisasue, N., "Large eddy simulation of vortex flow in intake channel of hydropower facility," Journal of Hydraulic Research, vol. 48, no. 4, pp. 415–427,2010, doi: 10.1080/00221686.2010.491644.

[16] Guo, Z. W., Chen, F., Wu, P. F., & Qian, Z. D., "Three-Dimensional Simulation of Air Entrainment in a Sump Pump," Journal of Hydraulic Engineering, vol. 143, no. 9,2017, doi: 10.1061/(ASCE)HY.1943-7900.0001321.

[17] Sugino, Y., Kawakita, K., & Matsui, J., "Numerical simulation on flow in pump sump with free surface," in *2014 ISFMFE - 6th International Symposium on Fluid Machinery and Fluid Engineering*, Institution of Engineering and Technology, 2014, pp. 130 (5 .)-130 (5 .). doi: 10.1049/cp.2014.1249.

[18] Lei, J., Tang, X., Li, X., & Guo, M., "Numerical study on various vortices in pump sump based on LBM-LES," IOP Conf Ser Earth Environ Sci, vol. 240, p. 072022,2019, doi: 10.1088/1755-1315/240/7/072022.

[19] Amin, A., Kim, B. H., Kim, C. G., & Lee, Y. H., "Numerical Analysis of Vortices Behavior in a Pump Sump," IOP Conf Ser Earth Environ Sci, vol. 240, p. 032020,2019, doi: 10.1088/1755-1315/240/3/032020.

[20] Yanase, S. … Takami, T., "Numerical study of air-entraining and submerged vortices in a pump sump," IOP Conf Ser Earth Environ Sci, vol. 240, p. 032001,2019, doi: 10.1088/1755-1315/240/3/032001.

[21] El Hady, M. A., "EFFECT OF SUMP GEOMETRY AND SUBMERGENCE DEPTH ON FREE SURFACE VORTEX FORMATION OF PUMP INFLOW," International Journal of Fluid Mechanics Research, vol. 50, no. 3, pp. 1–15,2023, doi: 10.1615/InterJFluidMechRes.2023045128.

[22] Kumar, R., Khare, R., & Singh, A., "Numerical Simulation of Flow Through Sump Pump," 2024, pp. 219–229. doi: 10.1007/978-981-99-5755-2_25.

[23] Arboleda, G. & El-Fadel, M., "Effects of Approach Flow Conditions on Pump Sump Design," Journal of Hydraulic Engineering, vol. 122, no. 9, pp. 489–494,1996, doi: 10.1061/(ASCE)0733-9429(1996)122:9(489).

[24] Okamura, T., Kamemoto, K., & Matsui, J., "CFD prediction and model experiment on suction vortices in pump sump," The 9th Asian International Conference on Fluid Machinery, pp. 1–10,2007, Accessed: Feb. 06, 2025. [Online]. Available: https://cir.nii.ac.jp/crid/1050863727635816960.bib?lang=en

[25] Bertrand, O., Loisel, P.-E., Girard, C., Demenet, P.-F., & Schaguene, J., "Three-Dimensional Numerical Modeling for Flow Analysis Inside a Pumping Station," 2018, pp. 885–897. doi: 10.1007/978-981-10-7218-5_63.