Journal of Mines, Metals and Fuels, *73(8): 2339-2359; 2025. DOI: 10.18311/jmmf/2025/49419*

Print ISSN : 0022-2755

Journal of Mines, Metals and Fuels

Contents available at: *www.informaticsjournals.co.in/index.php/jmmf*

Intelligent Combination of Sump Model Study with CFD Analysis

S. G. Joshi[1*], Shyam Shukla[2] and Virendra Bhojwani[1]

[1]*Department of Mechanical Engineering, MIT Art, Design and Technology University, Rajbaugh Loni Kalbhor, Pune - 412201, Maharashtra, India; sgjoshi059@gmail.com*

[2]*Department of Civil Engineering, MIT Academy of Engineering, Alandi Road, Pune - 412105, Maharashtra, India*

Abstract

This paper presents an innovative approach to optimising the design validation and construction timeline of a stormwater pumping station. Traditionally, validation of sump designs involves witnessed Froude scale model studies conducted by ANSI/HI 9.8 standards, requiring approximately 16 weeks for analysis and recommendations. Subsequent civil construction design and groundwork activities typically extend the timeline to about 40 weeks. In this project, a Computational Fluid Dynamics (CFD) analysis was introduced early to assess sump flow behaviour for both the original and modified designs. This proactive step, although not part of the original scope, allowed the civil construction design to begin ahead of physical model results, with later comparison confirming alignment between CFD and physical study findings. The strategy synchronised design completion with the end of the monsoon season, offering the contractor a continuous 8-month dry period for execution. This integration of CFD led to a time saving of nearly 13 weeks and reduced overall project costs, demonstrating the practical benefits of advanced simulation in infrastructure development.

Major Findings: *The intelligent integration of CFD analysis with Froude scale model testing significantly reduced the design validation timeline by approximately 13 weeks. The CFD simulations successfully identified critical hydraulic issues such as swirl and vortex formation, enabling early design refinements. This dual-approach ensured design compliance with HIS standards while optimising the overall construction schedule and cost-efficiency.*

Keywords: *CFD, Froude Scale Model Study, Hydraulic Performance, Pump Sump Design, Storm Water Pumping Station.*

1.0 Introduction

In the domain of hydraulic engineering, the design and validation of pump intake structures, particularly sump configurations, play a vital role in ensuring the reliability and operational efficiency of stormwater pumping stations. These systems are critical in flood-prone urban regions, where any hydraulic inefficiency can lead to severe disruptions, property damage, and threats to public safety. Traditional validation methodologies, such as physical sump model studies based on Froude scaling, are globally recognised for their accuracy and are mandated by standards such as ANSI/HI 9.8. However, despite their reliability, these witnessed model tests are time-intensive, often taking 16 weeks or more, which can delay the initiation of civil construction activities. To address the limitations associated with sequential validation and construction planning, this study explores the novel integration of Computational Fluid Dynamics (CFD) analysis with physical sump model testing. While CFD is not yet universally accepted as a standalone compliance tool for sump validation under current HIS guidelines, it offers valuable qualitative insights into flow behaviour. In this research, CFD simulations were performed in

**Author for correspondence*

parallel with the physical model study, enabling early detection of hydraulic anomalies such as vortices, swirl, and flow separation. The outcomes of the CFD analysis informed design modifications and facilitated the early commencement of civil construction and reinforcement detailing.

This integrated methodology not only preserved the robustness of traditional physical validation but also significantly reduced the overall project timeline, achieving a time saving of approximately 13 weeks. The approach provided a strategic advantage by synchronising indoor design work with seasonal constraints, allowing uninterrupted outdoor construction during the dry period. The present article elaborates on the experimental procedures, CFD methodology, design modifications, and validation outcomes, demonstrating the efficacy of this hybrid approach in enhancing project delivery for a large-scale stormwater pumping station. The efficient management of stormwater is crucial in urban infrastructure to mitigate flooding and environmental hazards, necessitating advanced engineering solutions for pumping station design and construction[1]. Traditionally, the design validation of storm water pumping stations, particularly the sump, relies heavily on physical model studies conducted according to ANSI/HI 9.8 standards, a process that is both time-consuming and resource-intensive[2]. These physical models, often Froude scale models, require significant time for setup, experimentation, data collection, and analysis, typically extending the design validation phase to approximately 16 weeks[3]. Following this validation, the subsequent civil construction design and groundwork activities further prolong the overall project timeline, often reaching around 40 weeks[4]. The integration of CFD analysis into the design process offers a transformative approach by enabling early assessment of sump flow behaviour, potentially streamlining the design validation phase and accelerating the entire construction timeline[5]. The conventional approach to stormwater pumping station design, which relies heavily on physical model studies, can be significantly enhanced by the incorporation of CFD[6]. CFD provides a detailed understanding of fluid behaviour within the sump, allowing engineers to identify and address potential issues such as vortex formation, uneven flow distribution, and air entrainment early in the design process[7]. By simulating various design scenarios and operating conditions, CFD enables engineers to optimise the sump geometry and pump arrangement, leading to improved hydraulic performance and reduced risk of operational problems[8]. The implementation of a CFD analysis early in the design phase allows for the assessment of the original and modified designs, offering a unique opportunity to refine the design before physical construction begins. This proactive step, although not initially part of the project's scope, proved invaluable by allowing the civil construction design to commence in advance of the physical model results, leading to significant time savings. Furthermore, the subsequent comparison between CFD results and physical study findings confirmed a strong alignment, validating the accuracy and reliability of the CFD analysis. The application of CFD methods has been proven effective in diverse fields, including aerospace, hydraulic engineering, and environmental pollution prevention, showcasing its versatility and applicability in complex engineering challenges[9]. The early detection and resolution of design flaws through CFD not only reduces the need for costly rework during construction but also minimises the risk of operational issues after commissioning[10]. By using numerical methods, CFD has proven its usefulness in both investigating and optimising processes, particularly in simulating flow behavior[11]. The ability of CFD to accurately predict flow patterns, pressure distributions, and other critical parameters allows engineers to make informed decisions regarding sump design, pump selection, and operational strategies[12]. The accuracy of CFD simulations is highly dependent on the quality of the computational model, the selection of appropriate turbulence models, and the proper implementation of boundary conditions[13]. CFD is critical across various engineering fields, offering practical guidelines for complex flow challenges, where it enhances mechanical system performance and aids in addressing nuclear safety concerns. This includes assessing wind pressure distribution resembling tornado-like swirling flow[14]. CFD can optimise the thermal performance of electronics[15]. The integration of CFD into the design process of wind turbines allows for system-level analysis, focusing not on a single component but on the turbine as a system, improving the turbine quality and time to market[16]. The use of CFD has been shown to reduce the time required to complete projects and improve the accuracy of results. In the case of the storm water pumping station, the integration of CFD led to a time saving of nearly 13 weeks

and reduced overall project costs, demonstrating the practical benefits of advanced simulation in infrastructure development. This highlights the potential for CFD to transform traditional engineering practices and drive innovation in the construction industry[17-19].

2.0 Background

2.1 BOOT Contracts

The Public Private Partnership (PPP) project model is often used for proposing and executing large infrastructural development projects like drinking water schemes for large cities or townships, or large-scale lift irrigation schemes. The turn-key contracts for these projects are awarded to the most competitive Engineering, Procurement Construction (EPC) contractor based on their bid on a Build, Own, Operate, Transfer (BOOT) basis. This is a win-win situation for all concerned. Since the entire responsibility of design, planning, procurement, construction, installation, operation, and maintenance rests on a single party, there is no scope for starting a blame game of blaming others for any shortfalls in expected performance parameters of the system. The public gets continuous and reliable service, the contractor gets appropriately paid for engineering the job well, and the Government agencies get the satisfaction of writing down the correct specifications and acceptance norms for the job, considering the present and future requirements of the masses.

2.2 The EPC Contractor and the Problems

The contractors have multiple responsibilities. They have to design a technically and technologically superior system to meet the requirements of the project as per specifications. They have to write down appropriate specifications for the components of the system, and procure all reliable and proven equipments at the minimum possible life cycle costs to keep the project costs under control and at the same time should not allow any time over runs; install, operate and maintain the equipments in the system to keep the running costs at estimated level such that, they can provide the services to public at the predetermined unit prices and still get their expected profit level. Any unexpected failure of the pumps and their components and system, and restoration of the services to achieve or bring back customer satisfaction, during the completion of the BOOT period or premature failure before handover, costs money, and it is a straight loss to the contractor. So, the contractor wants no failure at all. This does not happen easily. The required quality and reliability need to be built into the design of the system. To achieve this, they make use of all the latest experimental and analytical techniques at the design and procurement and execution stages.

Most of the problems in pumps and pumping systems arise due to less attention given to the suction arrangements of pumps in the pump house at the design stage. To keep the pumping system running continuously without failure of the components of the system, it is necessary to design the suction sump of the pump house, using all the design guidelines available in the textbooks and standards on Intake Structure design. Sometimes it is possible to follow all the design guidelines of textbooks and recommendations of standards while designing the suction sump, and sometimes some dimensions of the design need to be compromised due to site constraints, space constraints or financial constraints. In either of the situations, it is necessary to validate the design before finalising it. This is best done by a technique called Sump Flow analysis in a geometrically similar, smaller, Froude Scaler model". This technique not only shows detrimental flow conditions occurring in the original sump design but also allows experimental validation with correction devices to eliminate those detrimental effects permanently from that design.

3.0 EPC Contractor's anxieties

3.1 Design

EPC contractors can get a design of the sump made using textbook guidelines and recommendations of various intake structure design standards, keeping in view the performance parameters to be achieved and the site constraints. They are aware that even if the sump designer follows all the guidelines and recommendations, problems of a hydraulic nature do occur to some extent, and they need to be eliminated. So simultaneous elimination of problems and validation of sump design is possible in the process of "physical Froude scaler model studies" When it is not possible to follow all the design guide lines and recommendations and only follow them partially due to non-availability of sufficient space, time and finance, the

improvements to near perfection for the emerging sump design by scale model studies of sump becomes essential.

3.1.1 Execution

Sump designers are aware that solutions like "Flow dividing baffles", " Curved baffles for directing the flow"," Vertical column arrays for proper flow distribution", "Cruciform under suction bell to eliminate submerged vortex", " Back wall splitter" and sometimes "Cross under suction bell" to reduce and bring the swirl within acceptable limits, "False backwall" addition of "brest wall" in the pump chambers at proper distance from pump centerline will emerge singularly or in combinations as the outcome of the sump model test, for eliminating detrimental flow effects in the proposed sump design. Most of these devices are to be built on the sump floor, and hence their reinforcement needs to be designed along with that of the raft. Civil designers must have the data on these devices to start the reinforcement design of the raft. Only for a few devices like brestwall, backwall splitter, etc., their reinforcement needs to be designed along with that of structural columns above the raft. Till the time the contractors complete the actual laying work of the raft, they can wait for the details of these devices, above the raft level. To save on time, if the contractor constructs the pump house only based on design from guide book dimensions and without waiting for the completion and validation of design and afterwards finds that some devices like, cruciform under suction bell, flow diverting baffles, vertical column arrays etc. are required, as per sump model studies, they have to follow a cumbersome process of incorporating them in the already constructed raft. They have to chip out the concrete from the location where the device is to be added, to expose the steel reinforcements and weld additional reinforcements to suit the device to be added and redo the concreting for the device. This weakens the original reinforcement and the mother concrete of the raft.

3.1.2 Dilemma

To avoid the suggestions coming from the sump model study, if designers design the sump with all the devices like cruciform, backwall splitter, extended cross on all four sides of the cruciform, curved baffles at the transition between source water and the inlet canal, there will be two effects. One effect is an increase in the cost, and the other effect is that the pump sump floor will become cluttered with devices and will pose problems for cleaning the sump during yearly or periodic maintenance.

In 4 months of the rainy season, (Middle of June, July, August, September up to middle of October) when actual outdoor civil work at site is not possible, contractors would like to complete the indoor office work of design, analysis, validation and model study etc. and be ready with validated designs and civil construction drawings, to start the actual site work from middle of October onwards. This gives them a continuous uninterrupted stretch of dry spell of 8 months up to starting of the next rainy season (Middle of June of next year) to complete the construction of pump house (outdoor work) and take up the work inside the constructed pump house (in door work like erection, commissioning of electro mechanical equipments) in the next rainy season. These are some important milestones of efficiency for them.

If the contractors undertake the risk, they will limit it to the raft level itself. They would then have to settle for a process of taking up the civil construction design and reinforcement design only after the complete design is validated by a sump model study.

3.2 About Froude Scaler Sump Model Studies

"Flow studies on Froude scale model" is a very reliable validation technique of sump design. It has been in use worldwide for more than 150 years. Sump model study is a sequential process carried out by specialised laboratories under the guidance of highly experienced engineers. Extracting the sump outline geometry from the proposed civil construction drawings of the sump/ pump house, getting a confirmation on it for its perfect match from customer, deciding the scale ratio, based on achievement of minimum specified Reynold's number (minimum 6 x 10^4 for suction bell and minimum 1 x 10^5 for bell at the end of suction pipe for models) and Weber number (minimum 240) from model dimensions and preparing a test write up on model test circuit with slave pumps, sump with transparent back wall and transparent model pumps with vortometers fitted at proper locations (4 x d from suction bell lower edge where d is the throat diameter of suction bell)in them and arranged on suction piping of slave pumps, throttle valves and calibrated magnetic flow meters with (+)(-) 1.5% accuracy, arranged at proper straight distance pipe lengths (10 x D upstream and 5 x D downstream where D is the discharge pipe

size of slave pump) on discharge piping of slave pumps and matrix of combinations of working and stand by pumps and different water levels in the sump are some of the preliminary activities. This test write-up was then approved by the customer. This is the stage at which the customer sometimes adds some more objectives of the test over and above the normal objectives of a sump model study, namely, no surface or tail vortices stronger than Type 3 as defined in HIS, no swirl more than 5 degrees, no mass rotation, no stagnant zones etc. Such an approved write-up then serves as a Memorandum of Understanding, and all concerned parties follow it till the logical end of the exercise. Based on the approved write-up, the laboratory then constructs the model and transparent model pumps with vortometers fitted in them and makes the model test arrangement. During the running of the model, the customer witnesses the test, in which, presence of detrimental effects on sump flows is noticed. If those effects are unacceptable, then flow correction devices are discussed with the customer and the devices which can be erected or constructed at the site are finalised. The devices are then converted to model scale and then produced and fitted in the model, and again, model runs are taken to ensure that detrimental effects are brought down to acceptable limits or eliminated. Then the sump design with those devices is finalised. Thus, an experimentally validated sump design is finally used. There is a preparation of documents involved. Government agencies and their consultants are involved. A lot of back-and-forth movement of documents and manufacture of model pumps, installation of the test rig, and internal testing are involved. Inspection of dimensions, by laboratory officers, is involved, and sometimes witness inspection of dimensional measurements is also involved. Finding a convenient date and time for all experts to witness tests on the original design and then on the design modified with devices is involved, and essentially, this sequential procedure takes its own time. If a proper sump model study is a long, drawn, systematic process to be completed step by step, the clause of witnessed sump model test makes it even longer.

4.0 Time lines

On average, the time required for the process of validation of sump design by witnessed physical Froude scale sump model studies is about 4 months (16 weeks). Once the experimentally validated design is available, the contractors start the further process of civil construction design of the pump house. Reinforcement design up to the raft level and then for the superstructure of the pump house building takes about 3 months, and then the drawings are released for civil construction. The workforce and the construction equipment and machinery are mobilised to the site within one month (4 weeks), and within a reasonable time of 2 months (8 weeks), the pump house construction comes up to raft level or sump bottom level, ready for building the superstructure over it. This is a very long, drawn process, and if followed sequentially, the actual civil construction work can start approximately 8 months after the date of the proposal of the sump design.

4.1 Intelligent Combination

Authors were able to reduce this time for the EPC contractor to almost half without losing the assurance or backup provided by HIS to the sump design by using a novel method of combining CFD analysis of the prototype sump and physical study of its model sump. CFD was not a part of the specification. But it was added by the authors to reduce the overall time substantially. The two exercises done on the design of the sump of a storm water pumping station in a coastal city to bring home the point are described in this study. Since the laboratory had signed a non-disclosure agreement with the customer, the study does not mention any names related to the project. However, wherever essential, the authors have used fictitious names to hide the true identity.

5.0 Description of the Scheme

In this coastal city, normal drainage water occasionally mixes with storm water in the rainy season and collects at the outfall near Kinara Hotel and flows into the sea by gravity. During high tide in the sea, the sea water does not allow the incoming drainage water to flow by gravity and sometimes enters the drainage system itself in the reverse direction to flood or inundate the upstream areas of the city. The city's systems remain under water for quite some time and create havoc, and the normal life of thousands of citizens gets disrupted. There is property damage accompanied by loss of life. The installation of Storm Water Pumping Stations (SWPS) is a solution to these situations. In the design of SWPS, the drainage

water is directed into the sea by a gravity-guided channel. The inverse level of this channel is somewhere in between the lowest and the highest tide level of the sea. On this channel, mitre gates are installed. These gates operate automatically based on signals from level sensors placed on the inside and outside of these gates. When the sea water level is low, the gates are open and drainage mixed with storm water (if any) goes into the sea by gravity. When the sea water level is high during high tide, the gates close automatically and sea water is restricted from entering the drainage system in the reverse way. So, the drainage and stormwater start collecting in the sump of SWPS. In this sump, pumps designed to lift calculated flow capacity and throw it into to discharge channel are installed. This discharge channel is designed to throw water into the sea. The invert level of this discharge channel is well above the highest tide level of the sea, and hence, this pumped water falls into the sea by gravity. The number of pumps and their flow capacities are designed in such a way that even at the highest incoming flow of drainage and storm water on a day of unprecedented rains and highest water level in the sea due to high tide occurring simultaneously, the city area on the upstream side will not become water logged or inundated. This is the basic system of a SWPS. As the system deals with the lives of citizens, a lot of features to enhance the reliability of the system and features to enhance the safety of public lives are added to it. They can be enumerated as follows.

The mitre gates are installed in the structure created in the path of the gravity channel created for drainage, and the stormwater water to go into the sea. In case of emergencies, if repairs to these gates are required, there has to be a provision for isolation of these gates. So, on both sides of the gates, a provision for inserting stop log gates is designed. There is a provision to operate the gates manually by overriding the signals from the level sensors, in case of some emergencies. Since the incoming water to pumps is a mix of storm water and drainage, bar screens with continuously mechanically operated cleaning rakes are required to prevent debris from going into the pumps and blocking the flow passages of the impeller and casing. The pump discharge is made to fall into the discharge channel over a weir. The level of the crest of the weir (+28.25M) is kept well above the highest tide level. (+27.5 M) This design feature ensures that in no situation will the water from the sea come back into the sump from the discharge side. Isolation Sluice gates at the entry to each chamber, silt traps for each chamber, are some of the additional design features added for reliability. A distribution channel behind the screen to carry flow to all pumps in the event of partial blockage of the screen is one more feature added for the reliability of the system.

The pump operation is controlled by a Programmable Logic Controller. If one pump can balance the inflow and outflow to maintain a safe water level on the city side (suction side), then only one pump works. If the inflow is more than, additional pumps will operate, such that the flood water level on the city side always remains at the safe mark. The controller stops or starts a pump by keeping logic in memory that the number of starts and stops in an hour that a motor can withstand, as per the manufacturer of the pump motor set, is respected. Another rule the controller follows is that the total number of running hours clocked by each pump remains almost the same.

6.0 CFD Analysis for the System

6.1 Client-Supplied Data

Auto CAD drawing of proposed sump system with chamber details, minimum and maximum water levels, pump dimensional data and data on screens, minimum and maximum rates of storm water inflow.

6.2 Scope of the Work

It is from the sump inlets through the silt pit, screen, sluice gate, bell mouth to the pump outlet. The geometric model was constructed using the Pro-e CAD Modeler with the help of AutoCAD drawings provided by the client. Due to constrained space available, many dimensions in the overall sump design were required to be compromised and hence use of correction devices was envisaged.

6.3 Assumptions

It was a steady-state analysis, and flow obstructions due to the screen were modelled using a lumped system assumption based on the pressure drop across it.

6.4 Solver

This study utilised the industry standard CFD software package ANSYS CFX V15 with double precision to carry out flow simulations. The High-resolution advection

scheme is used in this exercise. In the case of recirculation of flows, this scheme gives good results since it is a higher-order scheme. The solution generally converged to a 10-4 RMS level. In the solution, the convergence of Mass and momentum is always ensured.

6.5 Turbulence Model

Based on previous experience, the Shear Stress Transport (SST) turbulence model was used in this study. Transport equations were solved for the turbulent kinetic energy (k) and specific dissipation rate (ω) in the SST turbulence model. This is a two-equation model. The SST model is k-ω based. It accounts for the transport of the turbulent shear stress, and hence it gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients.

6.6 Boundary Conditions

Appropriate boundary conditions are important for CFD analysis and its solution. The outer wall is considered a surface through which flow cannot pass, and at the surface, the velocity is zero. These walls are designated as walls with no slip. The total mass flow that is entering the sump is the inlet boundary condition and is applied at the entry. The boundary conditions in terms of pressure were specified at the outlet (Table 1). The free surface is considered a surface with zero normal gradients of flow properties. All the pumps are considered as pipes with thin surfaces and treated as walls with no slip. The domain type considered here is a fluid domain, and water is considered the flowing fluid in the domain.

Table 1. Boundary conditions

Sr. No.	Boundary Conditions	Values
1	Inlet flow rate	36m^3/sec
2	Outlet pressure	Atmospheric
3	Walls	No slip, smooth walls

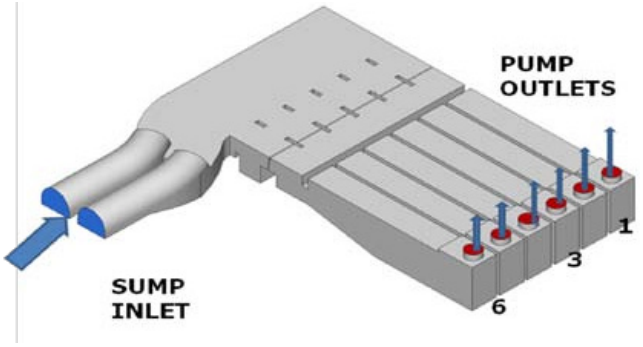


Figure 1. Initial 3D solid model- Actual size sump.

6.7 3D Geometry

Constructed from client client-supplied civil construction drawing

7.0 Grid Independence Study

It was carried out to see if the solution changes as the grid is refined. Finer and finer the grade, more is the number of nodes, and the computer time taken to converge on the solution and solve the problem becomes longer and longer. The requirement for computing capacity also becomes larger and larger. The cost of each iteration becomes higher and higher. In the Grid independence study, the same simulation is run on progressively finer grids (3 grids should be the minimum) by changing local and global grid parameters. At some finer grid, the solution becomes independent of the grid. The criteria for checking this are to maintain convergence of residual error to 10^{-4}, steady monitor points and imbalance below 1%. The purpose of this study is to prove that the solution finally obtained will be independent of mesh resolution.

7.1 Initial Grid Independence Study of 3 Meshed Sump Solid Models

A study with coarse, medium, fine and successively finer local and global mesh parameters was done (Table 2 and Figure 2). The purpose of this study was to see that

Table 2. Mesh information- Study 01

Sr. No.	Model	Elements	Nodes	ICEM Quality	Worst Skew	Aspect Ratio
01	Coarse	61,11,503	12,32,499	0.2679	0.732	1.977-13.182
02	Medium	76,43,773	15,34,396	0.3	0.7	1.347-13.47
03	Fine	93,34,087	18,58,082	0.2911	0.708	1.950-13.04

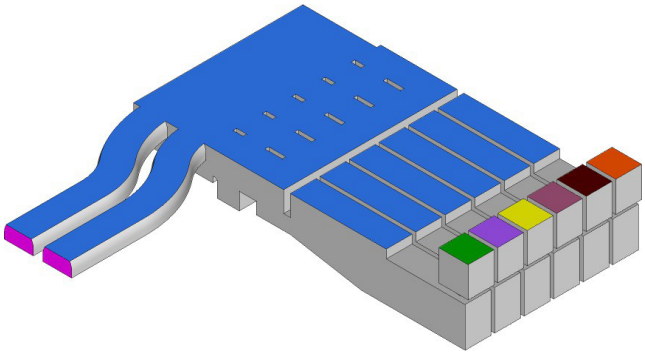


Figure 2. Model used for grid independence study.

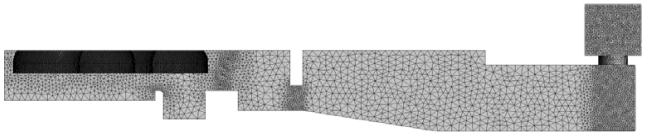


Figure 3. Coarse mesh 61L- Front view.

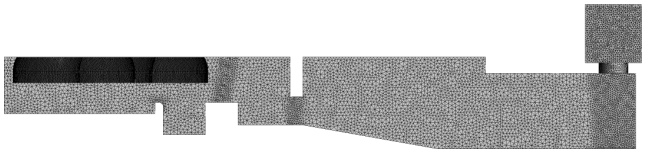


Figure 4. Medium mesh 76L- Front view.

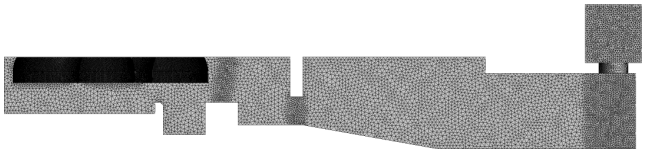


Figure 5. Fine mesh 93L- Front view.

the problem converges properly and that the solution is independent of the mesh resolution. The Sump solid model, mesh parameters and flow distribution are discussed below.

The following figures show three meshed models (Figures 3, 4 and 5), and the top views (6, 7 and 8) and front views (Figures 9, 10 and 11) of their streamline plots.

7.2 Discussions on Results of Initial Grid Independence Study

Flow rate through individual outlets was monitored to check the variation in results. Change in mesh. (Table 3) It can be seen clearly in Table 3 that there is variation in volumetric flow rate between all the models, so the solution, whether it is independent of mesh resolution

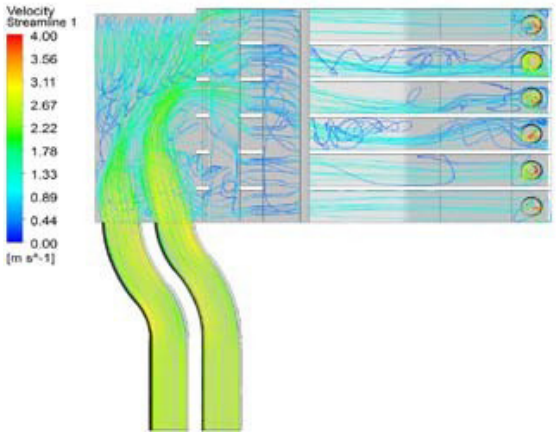


Figure 6. Streamlines- Top view - Coarse mesh.

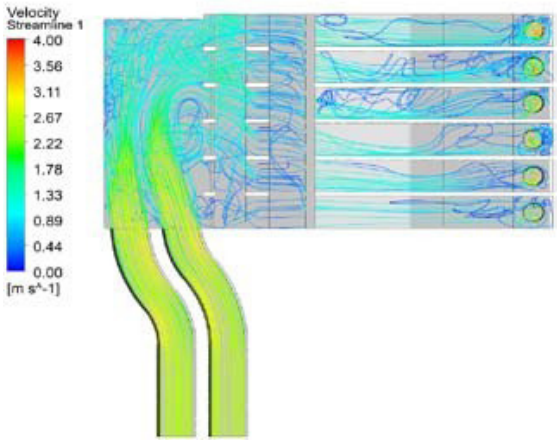


Figure 7. Streamlines- Top view - Medium mesh.

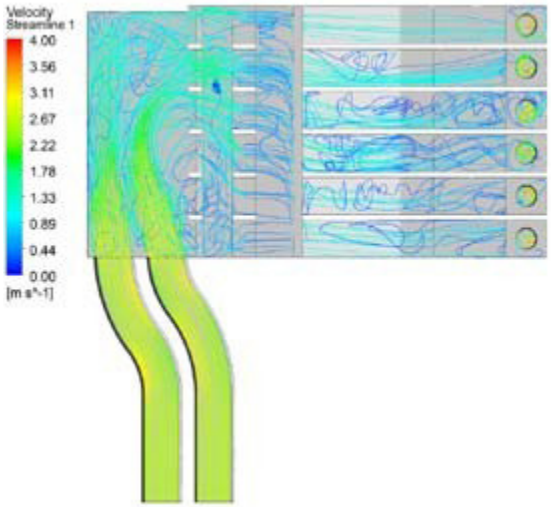


Figure 8. Streamlines Top view – Fine mesh.



Figure 9. Streamlines- Front view- Coarse mesh.

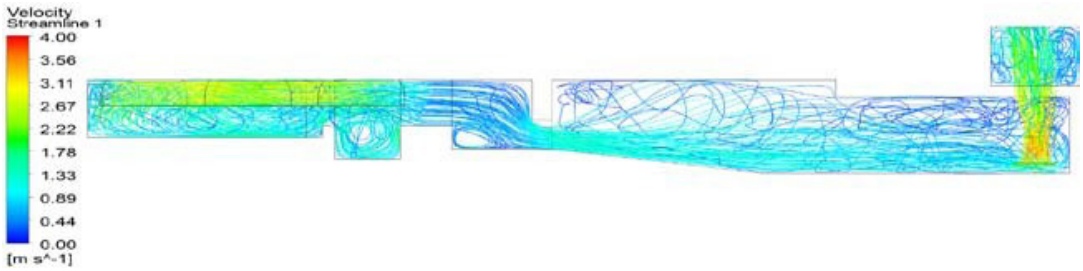


Figure 10. Streamlines- Front view- Medium mesh.

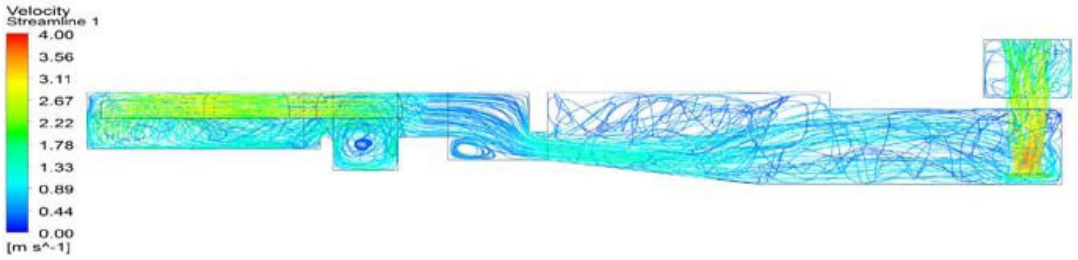


Figure 11. Streamlines- Front view- Fine mesh.

Table 3. Volumetric flow rate through outlets 1 to 6 for all 3 models

Model	Pump Out 01	Pump Out 02	Pump Out 03	Pump Out 04	Pump Out 05	Pump Out 06
Coarse	5.57	6.2	6.93	6.56	5.46	5.52
Med	5.38	6.35	6.6	6.45	6.12	5.32
Fine	5.75	6.11	6.57	6.11	6.38	5.38

or not, cannot be validated using only 3 mesh models. A larger number of models with different kinds of mesh parameters should be used to achieve grid independence.

8. New Grid Independence Study

The grid independence was further continued with 5 more meshed models as shown in Table 4.

The following figures show the 5 meshed models (Figures 12 to 16) with different mesh counts and top (Figures 17 to 21) and front view (Figures 22 to 26) of their streamline plots.

8.1 Discussions on Results of New Grid Independence Study

Flow rate through individual chambers was monitored to check the variation in results. change in mesh. Figures 27 and 28) and (Table 5) that there is negligible variation in volumetric flow rate between the mesh models 23L,

Table 4. Mesh information- Study 02

Sr. No.	Model	Elements (Tetras)	Worst Skew	Aspect Ratio
01	9L	932439	0.890	1.23-11.37
02	14L	1443965	0.890	1.23-10.55
03	23L	2301818	0.896	1.22-11.02
04	30L	3027077	0.896	1.22-9.59
05	40L	4054662	0.892	1.23-9.56

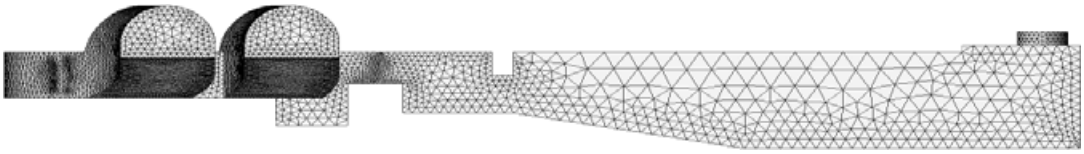


Figure 12. Mesh-Cell count 9L- Front view.

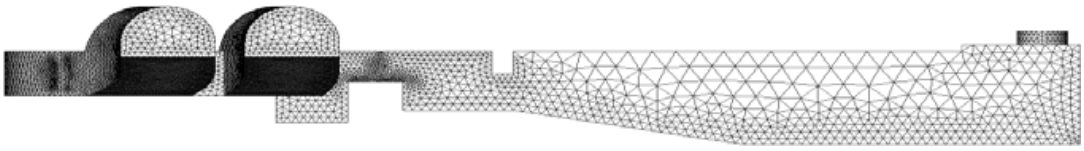


Figure 13. Mesh- cell count 14L-Front view.

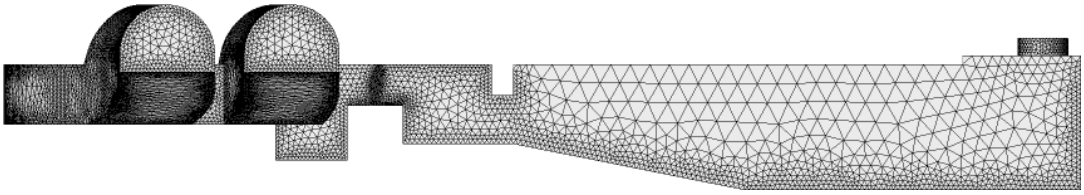


Figure 14. Mesh- Cell count 23L- Front view.

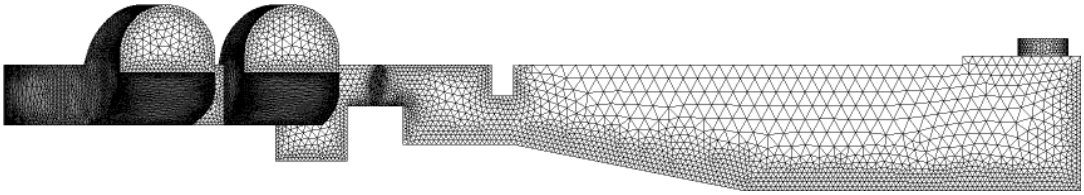


Figure 15. Mesh-Cell count 30L- Front view.

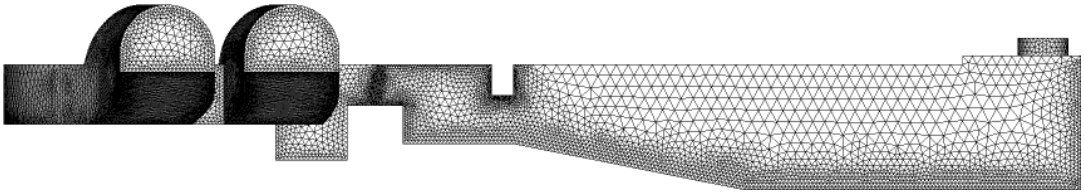


Figure 16. Mesh- Cell count 40L- Front View.

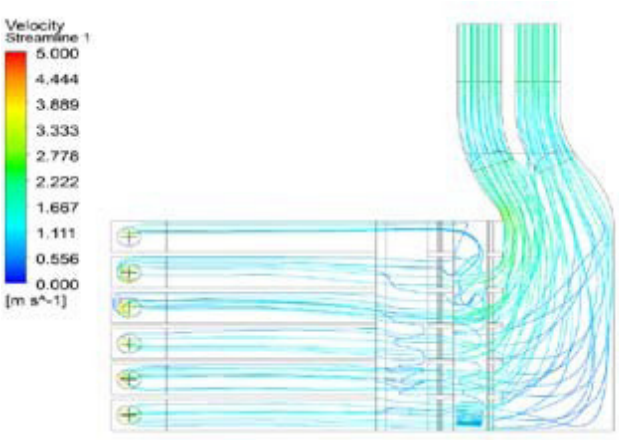


Figure 17. Streamlines Top view -9L mesh.

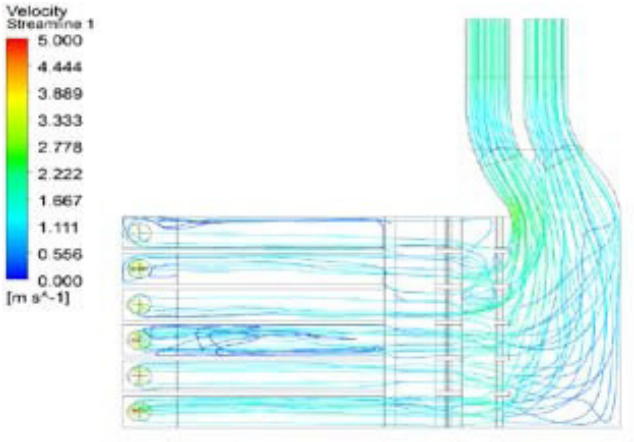


Figure 20. Streamlines top view -30L mesh.

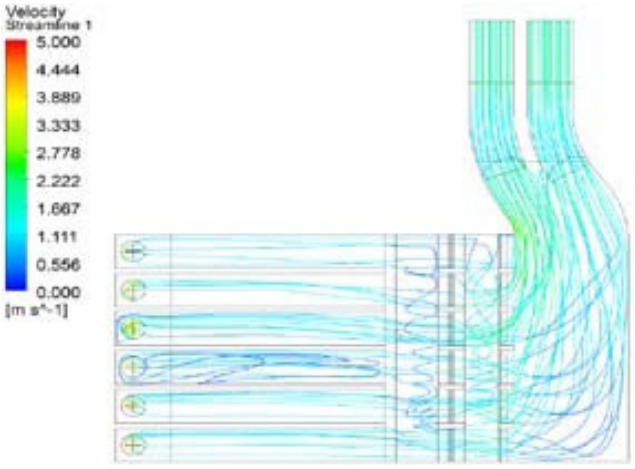


Figure 18. Streamlines top view -14L mesh.

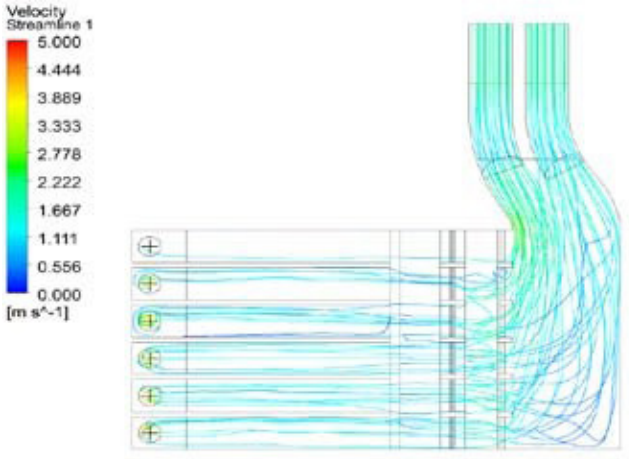


Figure 21. Streamlines top view - 40L mesh.

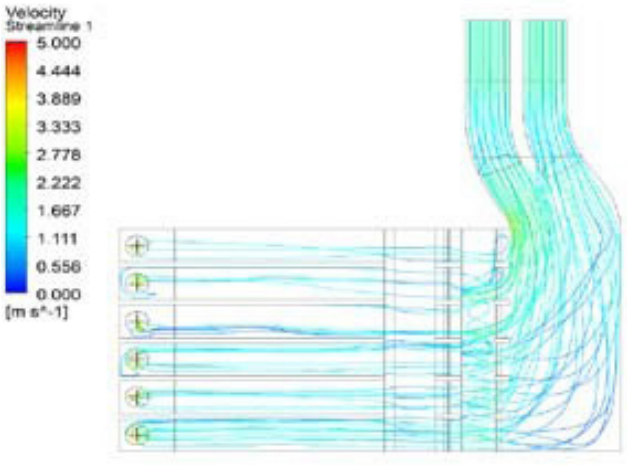


Figure 19. Streamlines top view -23L mesh.

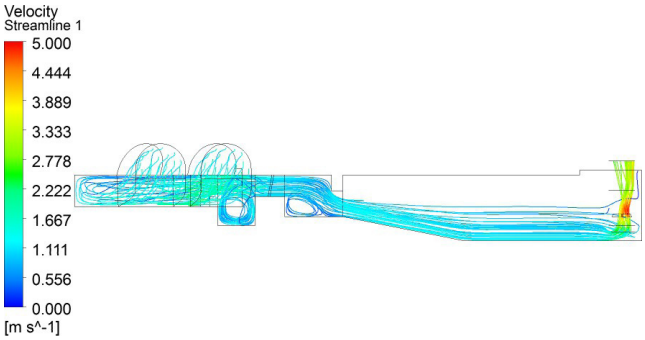


Figure 22. Streamlines- Front view- Mesh Model 9L.

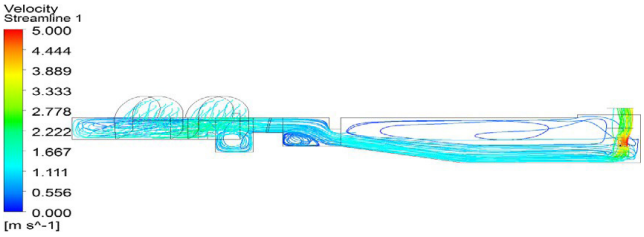


Figure 23. Streamlines front view- Mesh model 14L.

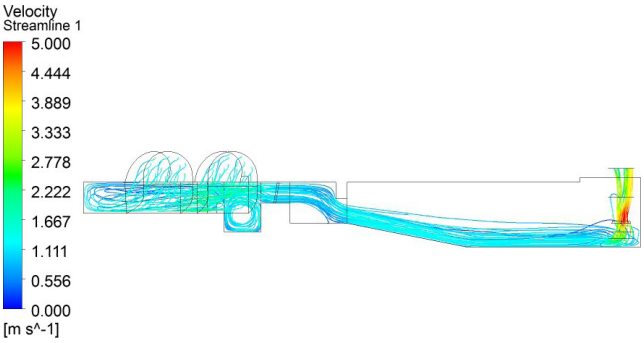


Figure 24. Streamlines- Front view- Mesh model 23L.

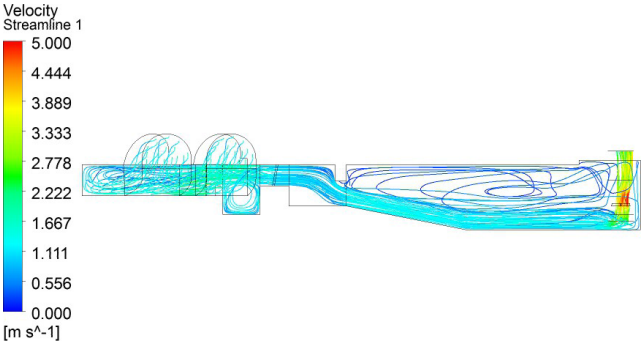


Figure 25. Streamlines- Front view- Mesh model 30L.

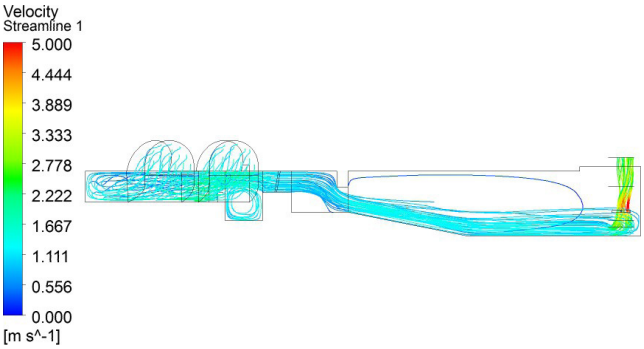


Figure 26. Streamlines- Front view- Mesh model 40L.

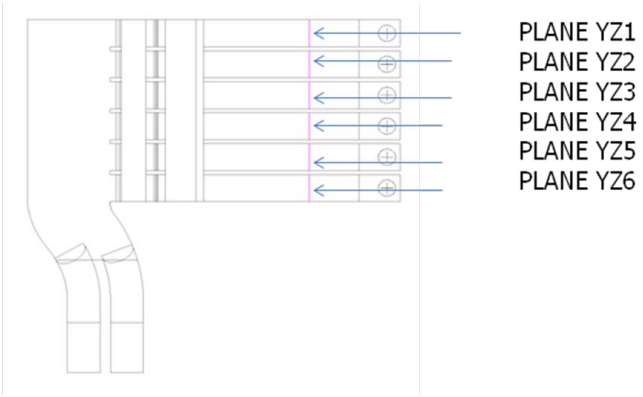


Figure 27. Locations of planes YZ (1 to 6) used for volumetric flow calculations.

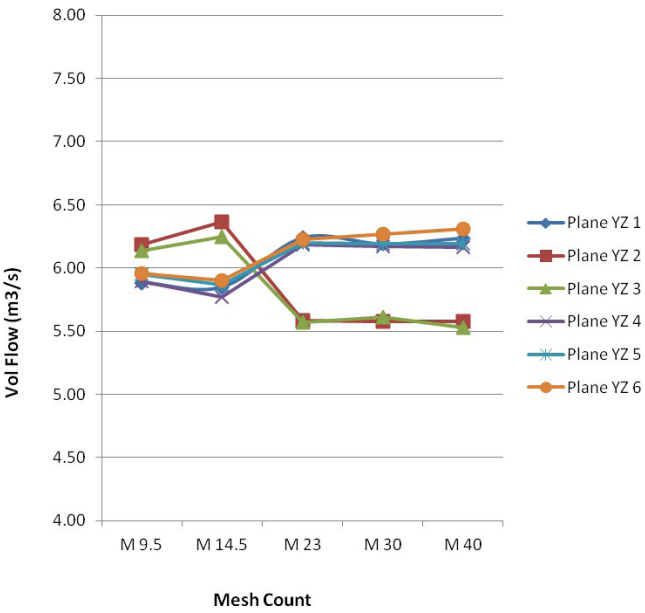


Figure 28. Volumetric flow rate compared for all 5 meshed models.

Table 5. Volumetric flow calculations through each pump

Location	9L	14L	23L	30L	40L
Plane YZ 1	5.88	5.85	6.24	6.19	6.23
Plane YZ 2	6.18	6.36	5.58	5.58	5.58
Plane YZ 3	6.13	6.25	5.57	5.61	5.53
Plane YZ 4	5.9	5.77	6.19	6.17	6.17
Plane YZ 5	5.95	5.87	6.2	6.19	6.19
Plane YZ 6	5.96	5.9	6.23	6.27	6.31

30L and 40L. Therefore, it is evident that the solution is independent of mesh resolution for 40L. So, in all further exercises, mesh 40L is used. It is only when the mesh became finer and finer, the presence of surface and submerged vortex (Figures 29 and 30) was noticed

9.0 Simulation

The number of iterations in the simulation process was dependent on the actual process of stormwater pumping that happens every day or every year. There is an incoming level of the drainage system at which the city is comfortable. This is the minimum water level in the sump. During rainy season when incoming flow increases many fold due to addition of storm water to drainage water and simultaneously high tide is there in the sea, the mitre gates will close and the pump sump starts getting filled by restricted inflow and one by one the pumps will start if the running pumps are unable to lift and throw away all the inflow in to the sea and water level in the sump starts rising. So, at the minimum water level, only one pump starts, and at the highest water level, all six pumps will be running. At the minimum water level, any of the 6 pumps could work. So, at the minimum water level, there will be 6 iterations with only 1 pump working and at the highest water level in the sump, which the city side allows as acceptable, all six pumps will be working. Thus, there will be only one iteration at the highest water level. These combinations were run in a simulation.

9.1 CFD Results (Original Design)

The presence of a submerged vortex under the suction bell and a surface vortex in the pump chamber near the pump canister and unacceptable swirl at the entrance, is noticed as shown in the extracts from post-processing streamline plots of the initial proposed design. Emergence of a submerged vortex was noticed under the suction bell of each pump when any single pump was in operation, when there was a minimum water level on the suction side. A submerged vortex was seen to emerge from the floor under the suction bell of all pumps, even when all six pumps were working at the highest water level in the sump. Surface vortex appeared and entered with a tail in the suction bell of every pump, when any one of the single pumps was operating at the minimum water level. When all six pumps operated at the highest water level, the surface vortices did occur, but their strength remained limited to slightly more than type 3, and their tail did

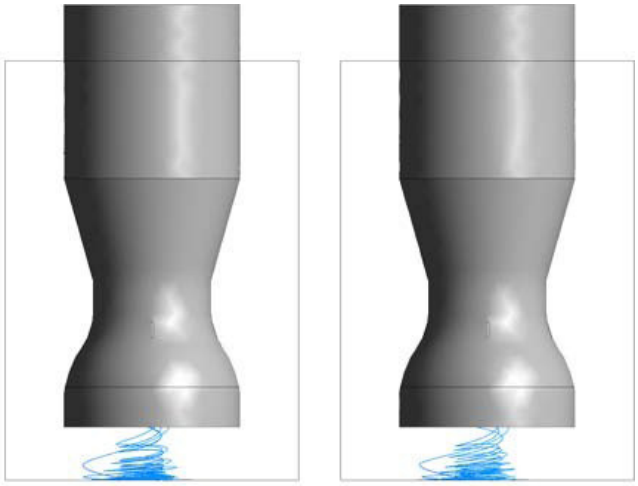


Figure 29. Presence of a submerged vortex under suction bell emanating from sump bottom.

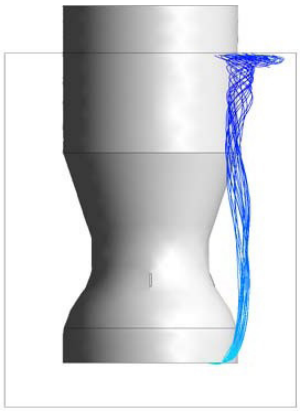


Figure 30. Presence of surface vortex (air entraining) Min. Water level

not reach up to the suction bells of the respective pumps. They diffused at a much higher level.

Suggested modifications (Figure 31, 32, 33 and 34) to eliminate detrimental effects in flow are as below

In addition a Brest wall at a distance of 2D from the pump centerline was added. It submerged 1.1m below minimum water level. It eliminated surface and air entraining vortex. The back wall splitter was retained to eliminate excessive swirl.

9.2 CFD Results After Simulation with Modifications

Post-processing results of simulations run with modification devices in place are shown in Figures 35, 36, and 37

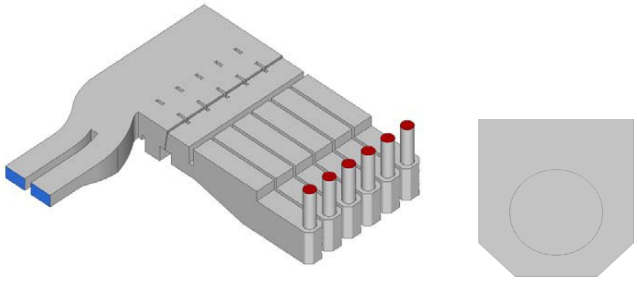


Figure 31. Chamfers at the back wall of chambers.

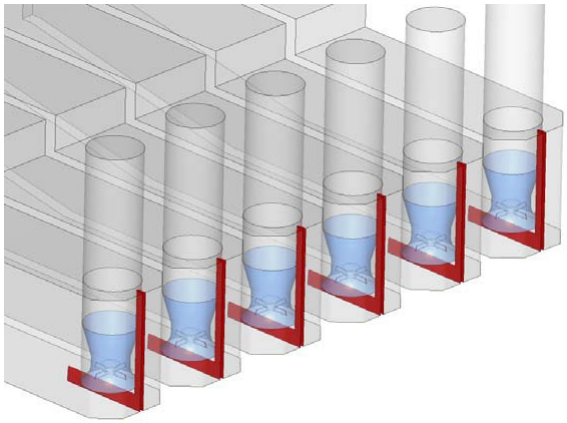


Figure 32. Backwall splitter and floor splitter tried as a solution.

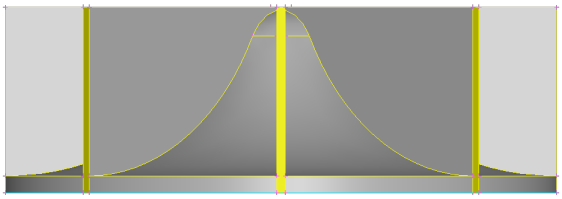


Figure 33. Details of the cruciform (4 bladed) under suction bell to eliminate submerged vortex.

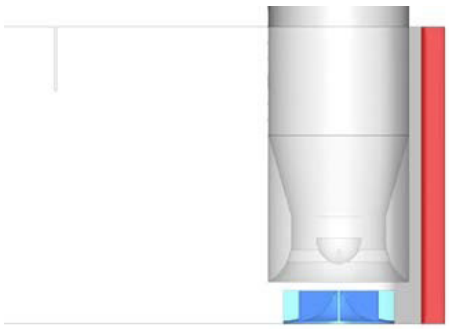


Figure 34. The floor splitter was replaced by a four-bladed cruciform in the final solution.

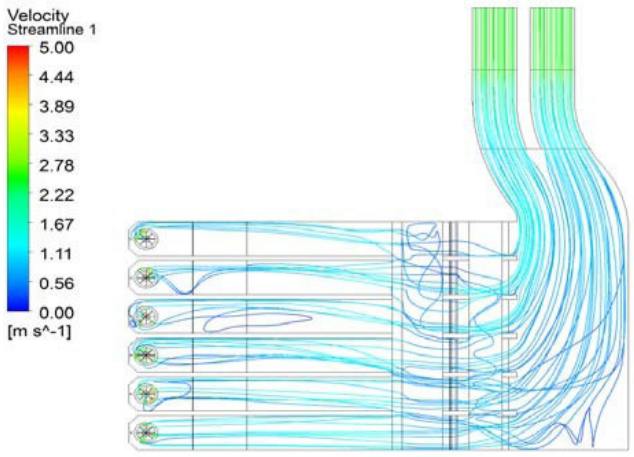


Figure 35. Streamlines- Top view- FINAL MODIFICATION.

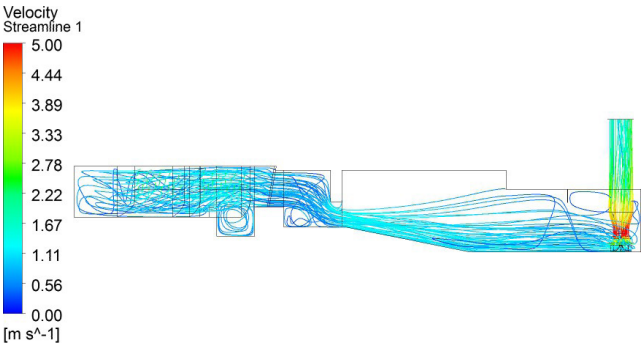


Figure 36. Streamlines- Front view- Final modification.



Figure 37. Velocity contour at plane after splitters- Final modification.

Values are acceptable since they are below 5 degrees (Results of each single pump in operation at minimum W.L. presented) (In one combined Table 6).

Table 6. Results of swirl angles at the pump entrance plane

Plane at pump entry	Pump 1	Pump 2	Pump 3	Pump 4	Pump 5	Pump 6
Swirl angle (degrees)	4.53	4.62	4.61	4.75	4.92	4.79

9.3 Final Activity

The drawings of various proven devices used (Figures 31, 32 ,33 and 34) were given to the contractor for starting the final civil construction drawings of the pump house complete with reinforcement. The time taken from readiness of preliminary proposed design drawing of pump house to getting a broad clearance for starting the civil construction drawing and the reinforcement drawing was exactly 22 working days.

10.0 Froude Scaler Physical Sump Model Studies as Per HIS

10.1 Client-Supplied Data

AutoCAD drawing of proposed sump system with chamber details, minimum and maximum water levels, pump dimensional data and data on screens, minimum and maximum rates of storm water inflow. This data is the same as that used for proceeding with the CFD analysis. Due to the constrained space available at the site, many dimensions in the overall sump design had to be compromised, and hence use of correction devices was envisaged during the sump model test.

10.2 Customer Requirement

The sump design of the proposed pump house should be validated using a physical Froude Scaler sump model study carried out as per clause 9.8.4 of HIS 9.8 of 2012 and acceptance criteria as per 9.8.4.7 of HIS 9.8 of 2012.

10.3 Objectives of the Test

Under any operating condition, of number of pumps working and any of the water levels either minimum or maximum at specified flow rate, the sump should not have vortex or air entraining vortex, and the water should not enter in to the pump with swirl angle beyond the acceptable level of 5 degrees and other detrimental effects like mass rotation, stagnation zones etc. should not be present. For acceptance criteria on vortex, table 9.8.4.5a of HIS 9.8 should be followed. Some important hydraulic aspects with respect to prototype and model pumps are shown in Table 7.

Selection of scale – 1:11 to achieve Re min. 60×10^4 & Wb min 240

Model to Prototype flow ratio = 1/ (scale ratio 11)$^{2.5}$
= 1/401.3

10.4 Model Construction (Reference drawing and photos, Figures 38, 39 and 42)

The model sump was constructed from M.S. Sheets, marine plywood and translucent acrylic sheets. The complete back wall portion of the sump was made from translucent acrylic sheets. The model pumps consist of a suction bell mouth and taper piece connected to a transparent acrylic suction pipe. A bladed vortometer (swirl meter) was provided in each pump. It was placed at a straight distance of 4 times the diameter of the pipes, representing the model pumps. The revolutions completed by the red blade of the vortometer in standard time of one minute (i.e. revolutions per minute) were taken as the measure of the extent of vorticity. This was visually observed with the help of a stopwatch. The swirl angle of maximum 5° was taken as an acceptable norm. In the collection tank, provision of perforated sheets/screens was made to kill the energy in the return flow and make it smooth and uniform.

Throttle valves and magnetic flow meters on the discharge side of slave pumps and return flow headers are seen in the foreground. Model pumps with vortometers and a translucent back wall of the pump house are also seen.

Each model pump is connected to the suction side of one slave pump. This is used for sucking the Froude scaler flow from each model pump. The flow rate from each pump is measured by passing it through a calibrated magnetic flow meter and is controlled by using a throttle valve in each discharge piping circuit. The discharge passing through the magnetic flow meter then goes back to a collection tank, which is connected to the model sump, as shown in Figures 40 and 41.

Table 7. Some important hydraulic aspects with respect to. prototype and model pump

Sr. No.	Description	For Prototype	For model	Remark	
1	Suction diameter Bell entry	Dp = 2100mm	Dm = 190.9 mm	1:11 scale	
2	Flow rate m^3/sec	Qp = 6m^3/sec, 6000 l/sec	Qm = 14.95l/sec 22.42 l/sec	1&1.5 model Froude Scaler flows	
3	Bell entrance velocity	1.733m/sec as per HIS guide	0.522 m/sec		
4	Reynold's number	-	113118	Model Min. required 6 x 10^4	
5	Weber number	-	665	Model Min. required 240	
6	Chamber width	3500mm	318 mm	Model required 300mm	Min.
7	Bell to floor clearance	700 mm D/3 as per HIS guide	63.6 mm	-	
8	Submergence below min. Water level	4250 mm	386 mm	Model Min. required 150 mm	
9	Back wall clearance	1575mm 0.75 D as per HIS guide	143.18 mm	-	
10	Column diameter	1969 mm	179 mm	Model required mm	Min. 80
11	Minimum. depth of water in the sump	Min W.L 23.2 – Sump bottom 18.25 = 4950mm	450 mm	-	
12	Chamber velocity	6/3.5 x 4.95 = 0.346m/sec	-	Max. as per HIS 0.5m/sec	
13	Screen Solidity ratio	35%	35%	Meeting HIS requirement	

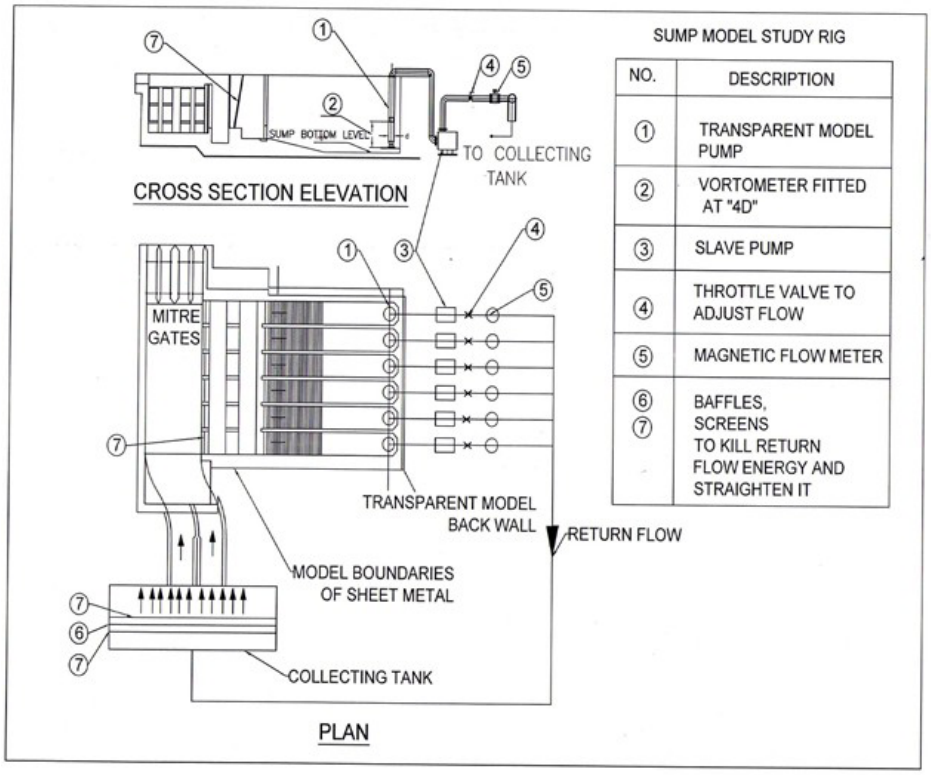


Figure 38. Overall Froude scaler (1:11) sump model test rig for the project.



Figure 39. 1:11 scale Froude scaler sump model test arrangement.

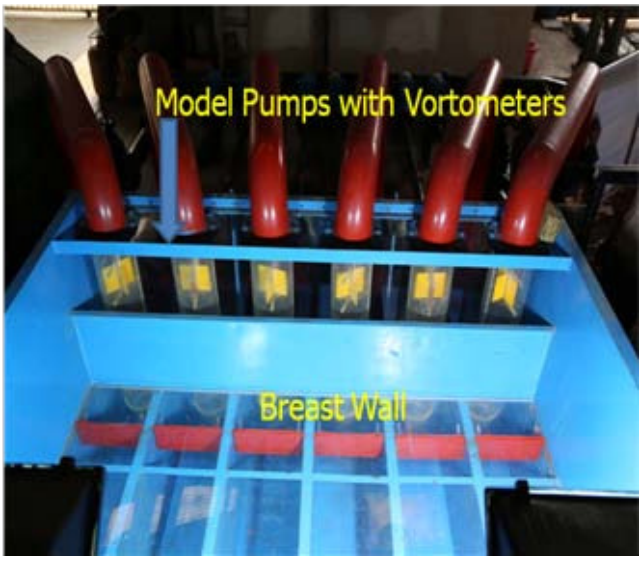


Figure 40. Model pumps with vortometers and brest wall and screens.

Different configurations/designs of model cruciforms tried are seen in the foreground on the right-hand side.

10.5 Test Procedure

At the minimum water level, one model pump was run at a time passing 1 Froude scaler flow and 1.5 Froude scaler flow through it each time and was repeated six times.

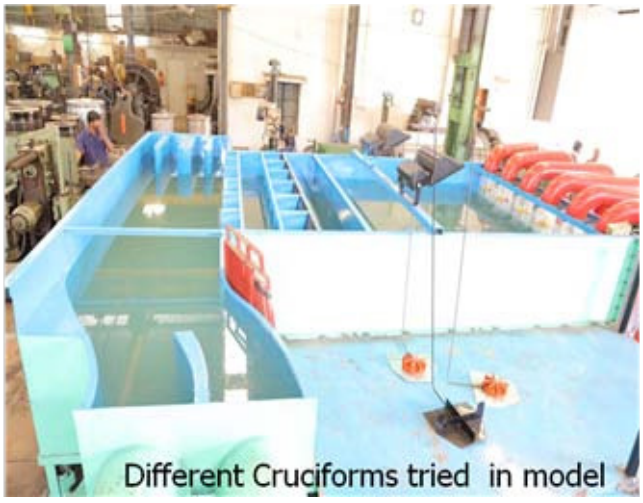


Figure 41. Another view of the 1:11 Froude scaler sump model test arrangement.

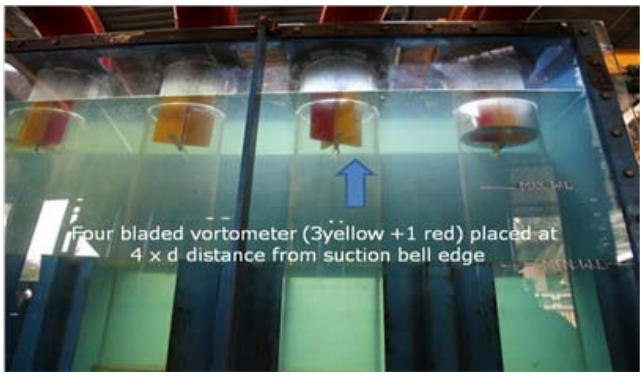


Figure 42. Four-bladed Vortometers (3 yellow and 1 red) are placed in each transparent pump model at a straight vertical distance of four throat diameters from bell throat.

The flow conditions in the chambers and on the surface of water in the forebay were observed visually, using dye injection and wool thread probe (Figures 42 to 46) under suction bell and were recorded photographically as well as videographically.

The vortometer revolutions were measured and recorded against a standard time of one minute with a stop watch, each time. At highest water level all six pumps were run and the readings were repeated and recorded.

10.6 Observations on Original and Modified Design

During the test, problems of submerged vortex emerging from sump bottom under suction bell of



Figure 43. Use of wool thread probe for noting flow direction and quality of flow.

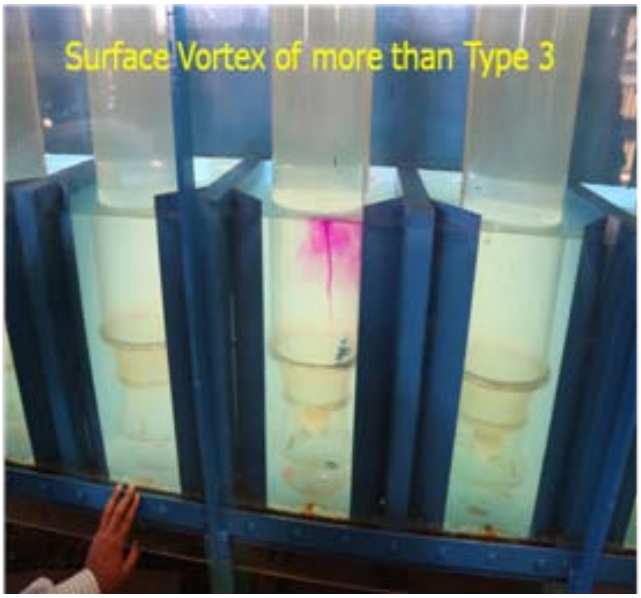


Figure 44. Vortex with tail stronger than type 3 seen in contrast due to dye injection.

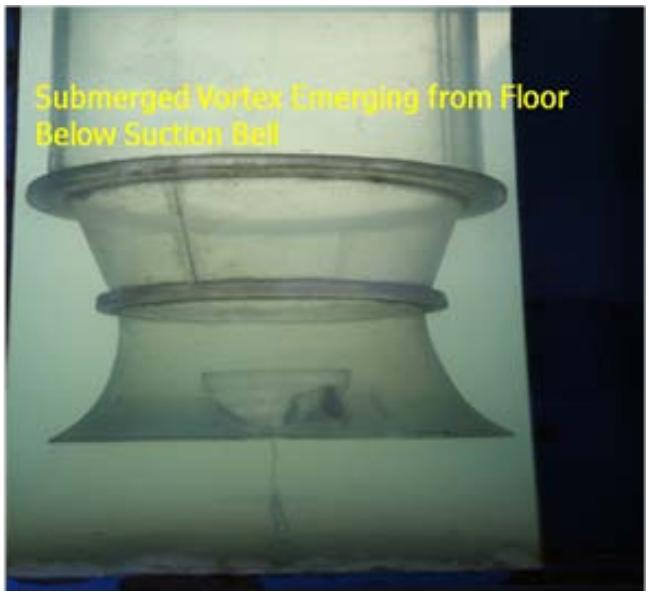


Figure 45. Submerged vortex.

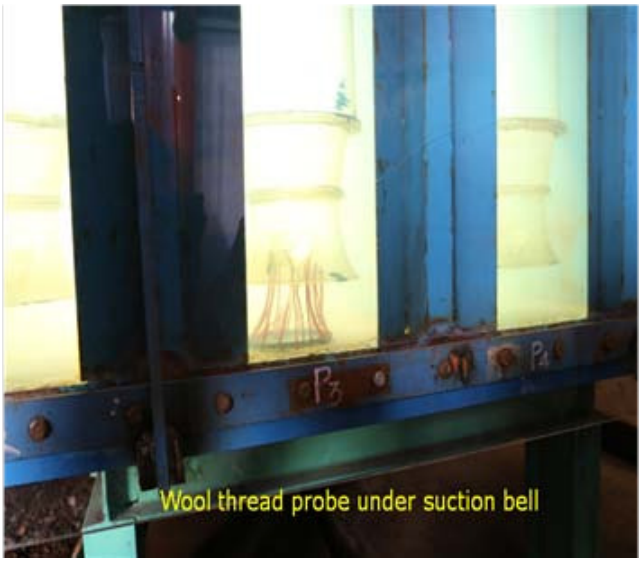


Figure 46. Wool thread probe under suction bell.

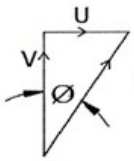
each pump, when operating singularly at minimum water level and all six pumps operating at highest water level, occurring of unacceptable swirl (very near to 5 degrees without any safety margin) at both minimum and maximum water levels in the sump and unacceptable surface vortex (stronger than type 3 as per HIS) at highest water level and air entraining surface vortex with tail going into suction bell at minimum water level were observed. After trial and error with devices, it was noted that the surface vortex got eliminated because of the breast wall, and because of the cruciform, the problem of submerged vortex got eliminated, and the swirl got reduced below the acceptable limit, due to the back wall splitter. Figure 47 shows observations at minimum water level and a 1.5 ft flow condition.

Observations at Min. water lavel and 1.5 F flow condition.

SR NO.	DISCRPTION	Swirl rpm by vortometer						REMARK
		P1	P2	P3	P4	P5	P6	
1	orignal sump	36	39	38	48	41	31	a)Submerged vortex from Sump Floor. b) Surface Vortex Air entraining tail entering suction bell.
2	modified sump	2	1	5	7	1.5	2	surface and submerged vortex Nil.sump performance found satisfactory

Note:- Modified Sump with 1).cruciformunder suction bell.
2).Vertical back wall splitter up to max water lavel.
3). Brest wall at distance of 530 mm from back wall and submerged 100 mm below min. water lavel.4)At Min.water each of the above readings are taken when only one pump was in operation at any given time.

Flow condition	Flow liter/sec.	Swirl rpm at swirl angle of				
		5°	4°	3°	2°	1°
1F	14.95	36.6	29	22	14.6	7
1.5F	22.43	55	44	33	22	11



V-AXIAL VELOCITY
U-PERIPHERAL VELOCITY

VELOCITY VECTOR DIAGRAM

Figure 47. Observations at the minimum water level and 1.5ft flow condition.

11.0 Conclusion

The objectives of this integrated exercise were multifaceted. The primary goal was to validate the sump design through a physical flow test using a geometrically scaled Froude model, in full compliance with clause 9.8.4 of the Hydraulic Institute Standards (HIS) 2012 and the corresponding acceptance criteria specified in clause 9.8.4.7. Although a prior CFD analysis had already indicated the necessity of flow correction devices, the physical model was initially tested in its unmodified form to assess inherent hydraulic deficiencies. The intention was to observe and confirm the presence of flow issues such as swirl, strong surface vortices, and submerged vortices at the lowest water level. Subsequently, the identified corrective devices were incorporated into the model, and the revised configuration was retested. This methodology successfully eliminated the observed detrimental effects and yielded a sump design that met the required validation standards within 16 weeks from the readiness of the initial design.

The second key objective was to leverage the monsoon season for indoor engineering activities, completing CFD analysis, validation processes, and the development of civil construction drawings, so that outdoor construction could commence immediately afterwards during the dry season. Since the CFD analysis was conducted as an internal activity without the need for third-party witnessing, it had a short lead time of approximately three weeks. When combined with the three-month civil construction design phase, it was possible to conclude all preliminary work within the monsoon window and initiate site activities by mid-October. This strategic planning allowed for uninterrupted pump house construction during the following eight-month dry spell. Although HIS guidelines emphasise that CFD alone cannot establish compliance without supporting evidence and validated acceptance criteria, in this context, CFD was used as a qualitative diagnostic tool. It provided early insights into potential hydraulic issues and enabled pre-emptive inclusion of corrective features into the civil design. This

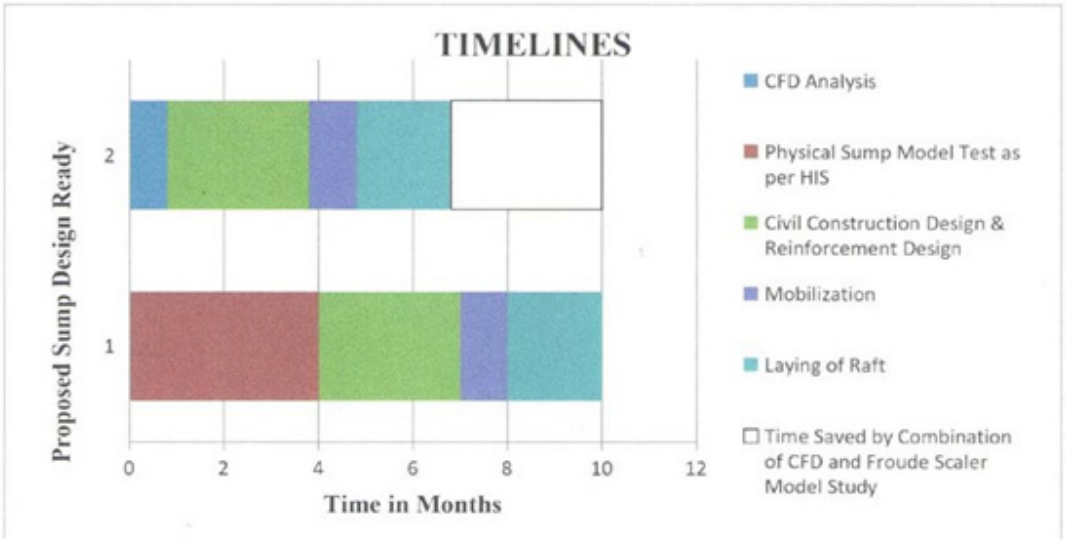


Figure 48. Depicting a clear-cut saving of 3 ¼ months due to the combination of CFD of the actual size sump and flow study in the Froude scale model (1:11) sump.

allowed reinforcement detailing and structural planning to begin just three weeks after finalising the initial sump design, offering a high level of confidence in eventual compliance with physical model study outcomes. Despite some quantitative differences, both CFD and physical model studies produced congruent findings regarding problem areas and viable solutions, allowing civil design to proceed without delay. Over the four months required for the physical model study, CFD analysis and civil design documentation were also completed. A comprehensive grid independence study ensured that CFD predictions were robust and representative, even though the finer grid used increased computational cost and time per iteration. Once confidence in the design alignment between CFD and physical validation was established, site mobilisation commenced, which itself required four weeks. By the end of the rainy season, conditions were optimal for starting raft construction based on the validated design. The final objective was to shorten the overall construction timeline for the sump and pump house. This was effectively achieved through the concurrent execution of CFD analysis and civil design during the period allocated for the physical model study, resulting in a time saving of approximately 13 weeks (3¼ months). This time efficiency also contributed to cost savings, making the combined validation approach both practical and economically advantageous.

The smp model studies, as per HIS, are an important part of specifications for validating the sump design. An intelligent combination of CFD analysis and Froude scaler sump model studies, both under expert guidance, can be used to save valuable lead time for the customer. Figure 48 shows a clear-cut saving of 3 ¼ months due to the combination of CFD of the actual size sump and the Flow study in the Froude scale model (1:11) sump.

12.0 References

1. Palaka R, Reddy PS, Viharika G, Pravalika G. Design of urban storm water drainage system using GIS and SWMM Software. In IOP Conference Series: Mater Sci Eng 2021; 1197(1):012014. IOP Publishing. https://doi.org/10.1088/1757-899X/1197/1/012014
2. Wang WL, Guo JC, Li JQ. Hybrid drainage design for highway underpass. J Irrig Drain Eng. 2020; 146(10):04020031. https://doi.org/10.1061/(ASCE)IR.1943-4774.0001501
3. Bisht DS, Chatterjee C, Kalakoti S, Upadhyay P, Sahoo M, Panda A. Modelling urban floods and drainage using SWMM and MIKE URBAN: A case study. Nat Hazards. 2016; 84:749-776. https://doi.org/10.1007/s11069-016-2455-1
4. Fan Y, Ao T, Yu H, Huang G, Li X. A coupled 1D-2D hydrodynamic model for urban flood inundation.

Adv Meteorol. 2017; 2017(1):2819308. https://doi.org/10.1155/2017/2819308
5. Nanía LS, Leon AS, García MH. Hydrologic-hydraulic model for simulating dual drainage and flooding in urban areas: Application to a catchment in the metropolitan area of Chicago. J Hydrol Eng. 2015; 20(5):04014071. https://doi.org/10.1061/(ASCE)HE.1943-5584.0001080
6. Gormley M. Can numerical computer modelling aid innovation, efficiency and cost reduction in sanitation provision? Desalination. 2009; 248(1-3):286-290. https://doi.org/10.1016/j.desal.2008.05.067
7. Smith LS, Liang Q, Quinn PF. Towards a hydrodynamic modelling framework appropriate for applications in urban flood assessment and mitigation using heterogeneous computing. Urban Water J. 2015; 12(1):67-78. https://doi.org/10.1080/1573062X.2014.938763
8. Xi Y, Zhang YL, Zhang XL, Dai Y. Enhancement of particle collection efficiency considering the structural interplay: Particle motion characteristics analysis. Mech Ind. 2020; 21(6):618. https://doi.org/10.1051/meca/2020093
9. Li Y, Luo Y. Application of computational fluid dynamics in groundwater pollution remediation. In J Phys Conf Ser. 2019; 1300(1):012070. IOP Publishing. https://doi.org/10.1088/1742-6596/1300/1/012070
10. Karnakanti S. Parametric study and simulation of horizontal split casing pump by using computational fluid dynamics.
11. Kurhade AS, Bhambare PS, Desai VP, Murali G, Yadav RS, Patil P, Gadekar T, Biradar R, Kirpekar S, Charwad GA, Waware SY. Investigating the effect of heat transfer influenced by the application of wavy corrugated twisted tape inserts in double pipe heat exchangers. J Adv Res Fluid Mech Therm Sc. 2024; 122(2):146-155. https://doi.org/10.37934/arfmts.122.2.146155
12. Belhadj E, Chilton S, Nimmo W, Roth H, Pourkashanian M. Numerical simulation and experimental validation of the hydrodynamics in a 350 kW bubbling fluidised bed combustor. Int J Energy Environ Eng. 2016; 27-35. https://doi.org/10.1007/s40095-015-0199-4
13. Hanspal N, Cryer SA. The use of Computational Fluid Dynamics (CFD) within the agricultural industry to address general and manufacturing problems. Fluids. 2024; 9(8):186. https://doi.org/10.3390/fluids9080186
14. Makhkamova I, Mahkamov K, Taylor P. CFD thermal modelling of Lynx overhead conductors in distribution networks with integrated renewable energy driven generators. Applied Thermal Engineering. 2013; 58(1-2):522-535. https://doi.org/10.1016/j.applthermaleng.2013.04.066
15. Kurhade AS, Murali G, Jadhav PA, Bhambare PS, Waware SY, Gadekar T, Yadav RS, Biradar R, Patil P. Performance analysis of corrugated twisted tape inserts for heat transfer augmentation. J Adv Res Fluid Mech Therm Sci. 2024;121(2):192-200. https://doi.org/10.37934/arfmts.121.2.192200
16. Kataoka H, Ono Y, Enoki K. Applications and prospects of CFD for wind engineering fields. Journal of Wind Eng Ind Aerodyn. 2020; 205:104310. https://doi.org/10.1016/j.jweia.2020.104310
17. Wang Y, Wong KK, Du H, Qing J, Tu J. Design configuration for a higher efficiency air conditioning system in large space building. Energ Buildings. 2014; 72:167-176. https://doi.org/10.1016/j.enbuild.2013.12.041
18. Arnold M, Wenz F, Kühn T, Lutz T, Altmikus A. Integration of system-level CFD simulations into the development process of wind turbine prototypes. J Phys Conf Ser. 2020; 1618(5):052007. IOP Publishing. https://doi.org/10.1088/1742-6596/1618/5/052007
19. Veynante D, Vervisch L. Turbulent combustion modelling. Progress in Energy and Combustion Science. 2002; 28(3):193-266. https://doi.org/10.1016/S0360-1285(01)00017-X