CFD ANALYSIS OF VORTICES AT THE SUCTION PIPE

M.TECH PROJECT REPORT

Submitted By

ADITI RAWAT

(Roll no. 2K15/HFE/02)

Under the Guidance of

Prof. Bharat Jhamnani

DEPARTMENT OF CIVIL ENGINEERING

DELHI TECHNOLOGICAL UNIVERSITY

DELHI-110042

(July,2017)

CANDIDATE’S DECLARATION

I do hereby certify that the work presented is the report entitled “**CFD ANALYSIS OF VORTICES AT THE SUCTION PIPE**” in the partial fulfilment of the requirements for the award of the degree of “Master of Technology” in Hydraulics & Water Resources Engineering submitted in the Department of Civil Engineering, Delhi Technological University, is an authentic record of our own work carried out under the supervision of Mr. Bharat Jhamnani (Assistant Professor), Department of Civil Engineering.

I have not submitted the matter embodied in the report for the award of any other degree or diploma.

Aditi Rawat

Date: (2K15/HFE/02)

CERTIFICATE

This is to certify that above statement made by the candidate is correct to best of my knowledge.

Mr. Bharat Jhamnani
(Assistant Professor)
Department of Civil Engineering
Delhi Technological University
Delhi

ACKNOWLEDGEMENT

I take this opportunity to express my profound gratitude and deep regards to Prof. Bharat Jhamnani, Department of Civil Engineering, DTU for the constant encouragement, guidance and support throughout the course of this project work.

I would like to thank all faculty members of Civil Engineering Department for extending their support and guidance.

I express my sincere thanks to all my colleagues and seniors for their help. I thank my parents for their love, care and moral support.

ADITI RAWAT
(2K15/HFE/02)

ABSTRACT

The primary goal of this study is to comprehend the occurrence and characteristics of vortices around the suction pipe of a pump sump system. The study is carried out by conducting numerical studies using CFD (Computational Fluid Dynamics). This study is done in two phases, in first phase, comparison of three CFD models i.e. standard k-Ɛ model, reliable k-Ɛ model and shear stress transport model is conducted, where in second phase comparison on occurrence and characteristics of vortices for different submergence depth i.e. 300mm, 130mm and 50 mm are done. For the first case, model is scaled down to 1:10 and considered one suction pipe of a sump chamber and for the second case, model is scaled down to 1:12 which consist three suction pipe and a sump. The model comparison case suggested that shear stress transport model is a better model than the k-Ɛ models, but in between k-Ɛ models, standard k-Ɛ model is better than the reliable k-Ɛ model. This comparison is done on the basis of accumulated time step, volume fraction of water, vorticity, pressure distribution and turbulent kinetic and dissipation energy, where there maximum and minimum values were compared according to their occurrence on time step. For the submergence depth case, comparison of characteristics of vortices and swirl angle calculation are carried out. Swirl angle at all three suction pipe is calculated and compared for all three different submergence depth. Swirl angle for suction pipe 2 for depth 300mm, 130mm and 50mm are 3º, 6º and 8º, where it can be said that swirl angle is decreasing with the increase in submergence depth. According to HI standard 1998, the permitted maximum swirl angle is 5º, so with reference to HI standards CFD analysis produced desirable results for suction pipe 2 at 300mm depth. But when studied as a system, there is decrease in swirl angle with increase in submergence depth, but, the swirl angle is not less than 5º. There is a validation of model of scale 1:12 is done with the obtained result from a journal of Korea university[1], for swirl angle and location of vortex core region. The study in this report helps to visualize the occurrence, location and time variation of vortices around the suction pipe and the effect of change in submergence depth on vortices and characteristics of vortices.

Keywords : CFD models, vortices, swirl angle, submergence depth

Table of Contents

List of Figures

List of Tables

List of Graphs

Symbols and Abbreviations

S.no	Description	Symbol
1	Density	ρ
2	Viscosity	μ
3	Kinematic viscosity	ϑ
4	Acceleration due to gravity	g
5	Froude no	Fr
6	Shear stress	τ
7	Pressure	p
8	Velocity	v
9	Circulation	ω
10	Surface tension	σ
11	Submergence	S
12	Energy	e

CFD – Computational Fluid Dynamics

HIS – Hydraulic Institute Standards

TKE – Turbulent Kinetic Energy

CHAPTER-1

INTRODUCTION

1.1 GENERAL

A suction pipe of a pump sump system is used to lift water that is collected in a sump basin to the pump intake via a phenomenon of suction in which fluid sucked into a partial vacuum or region of low pressure. The principle point of suction pipe is to intake water with uniform speed at the time of pump operation because the entrance of non-uniform flow causes phenomena such as vortices, air entrainment, pressure loss, cavitation, flow separation, vibration and noise.

Electrical power generating plants, cooling system, lift irrigation, water supply, sewerage pumping system, thermal power plant etc. are some of the many application where any defect in pumping operation can cause serious consequences. Vortices, cavitation, spiral flows and reverse flows evolving at the suction pipe of the pump are the causes for the disturbance and the low efficiency of pump's operation.

Pumps consist of a channel where the fluid enters the sump and then through the suction pipe to the pump and an outlet where the fluid fall out. The entrance of the conveyance is inlet that said to be the suction side of the pump. The outlet area said to be discharge side of the pump, where fluid falls out. Operation of the pump starts from suction side (lower pressure) with the goal that fluid can enter suction pipe to the intake of the pump. Pump operation cause higher pressure at the release side by forcing liquid at the outlet. The vortex development at the pump intake is a typical issue experienced at the time of pump operation. The vortices are developed due to the low submergence and high intake velocity, which prompts mechanical damage at impeller and loss of pump execution. Vortices are formed where fluid flow rotates around an axis. To characterize vortices terms like velocity, vorticity and circulation are consider. Vortices are of two types one is irrotational vortex and other is rotational vortex. With a goal to meet the normal flow, in some cases different pumping frameworks are utilized where suction pipes are found adjacent. In view of this course of action, the execution of the pumps get impacted.

Sump pump frameworks are used in modern and business applications to control water table-related issues in surface soil. Pump sump assumes an imperative part in the cooling water frameworks of thermal power plants. They are likewise utilized as a part of enormous lift water system, drinking water units for the supply of water. An artesian aquifer or intermittent high water table circumstance can make the ground end up unsuitable and damage because of water saturation. For whatever length of time that the pump work, the surface soil will stay stable. These sumps are for the most part 3cm top to bottom or more fixed with layered metal pipe that contains holes or deplete gaps all through. Vertical Sump Pumps are utilized as a part of the industrial pumping applications to pump clean, fluids consist of large solids and fibrous slurries from the deep sumps. The pumping head is suspended or submerged into the fluids and the drive motor is arranged above.

Due to these wide variety of applications, plan of suction pipe for pump sumps has picked up a considerable importance. A broadly acknowledged arrangement of rules created by the Hydraulic Institute Standards (1998) are frequently used to help the engineers in the design of these pump sumps. Since no single arrangement of rules can fulfill all the possible situations, a design require a physical hydraulic model examination, which can be led to guarantee adequate flow conditions if the plan goes astray from standards or exceeds maximum flow rate.

1.2 PHYSICAL MODEL STUDY

The physical model study considers flow and velocity estimations and flow visualization. In the event that the flow characteristics are not correctly, acquired, suitable measures can be taken. To diminish these vortices and for the better suction pipe design for pump sump efficiency, it is necessary to understand detailed flow data for sump framework including suction pipe. For this reason, numerous analysts are doing experiments and numerical investigations on flow around suction pipe of sump pump.

The phenomena of Hydraulic that influence flow around the suction pipe are as below:-

- Excessive pre swirl of flow entering the pump
- Entrained air or gas bubbles.
- Non-uniform spatial distribution of velocity at impeller pipe.
- Submerged vortices.
- Excessive variation in velocity and swirl with time.
- Free Surface vortices

The negative effect of each of these phenomena around suction pipe and on pump execution relies on minimum submergence depth, specific speed and size of a pump, and also other design components of pump. In general, high specific speed pumps are sensitive to adverse pump phenomena.

Components of adverse hydraulic powered conditions are

- ❖ Reduced flow rate
- ❖ Reduction in developed head
- ❖ Increased power consumption
- ❖ Increased vibration and noise

These unfavorable conditions can be avoided by proper model investigations and the efficiency of flow condition in the model can be corrected by adding few structures like baffle plates, guide vanes, flow splitters, ramps etc. for the development. Since model investigations in lab takes lot of time and money, now days numerical investigations are being done with the utilization of computational fluid dynamics.

1.3 NUMERICAL MODEL STUDY

Numerical models can replace the experimental set ups which are very costly and time taking. Despite the fact that they can't replace the experiments totally, the scale of cost and time in experimentation can be decreased to a specific degree.

Once a numerical model is set up, appropriate design boundary conditions are applied. In numerical models, we can apply those conditions that are hard to apply during experiments. In any case, we can't totally depend on the numerical outcomes, due to the deficiency of the scientific model i.e. accepting ideal conditions. Because of the limitations in the accessible computing power, the quality of simulations is influenced. Utilization of Computational Fluid Dynamics in numerical simulation has expanded. Other programming softwares which are in use are FLUENT, PHEONIX, OPEN FOAM, CFX.

1.4 COMPUTATIONAL FLUID DYNAMICS

Numerical model studies of fluid mechanic and hydraulics are carried out in Computational Fluid Dynamics (CFD). It consist of different model for every small and complex situations. CFD softwares is used to perform huge number of calculations to simulate the interactions of liquids and gases with surfaces that are characterized by boundary conditions.

The premise of all of the CFD problems are Navier Stokes equations that signify degree for (gasoline or liquid) fluid flow. The method considers the development of a version, indicating the boundary conditions, doing the simulation and post processing. Finite volume method is the most widely taken into consideration approach to solve in CFD codes in view of its gain in memory use and velocity mainly for large problems like high Reynolds wide variety turbulent flows and supply dominated flows (combustion).

1.5 OBJECTIVES OF THE STUDY

The principle goal of the present study is to research on the flow characteristics of vortices around the suction pipe of a sump pump for different submergence depth at 300mm, 130mm and 50mm; by carrying out numerical simulation in CFD on the scaled models and by visualizing the flow phenomena, incompatible hydraulic conditions influencing the flow and to recommend remedial measures. In this study, model tests on CFD are done to get velocity values and swirl angles. The objective of the study includes:

1. To investigate different numerical model i.e. standard k-Ɛ model, reliable k-Ɛ model and shear stress transport model; and to find out more suitable and effective numerical model for simulating vortex around the suction pipe with Volume of fraction method considering accumulated time steps.
2. To perform numerical simulation on scaled models to study occurrence and characteristics of vortices around the suction pipe for different submergence depth. Comparison of tangential velocity, radial velocity, turbulent kinetic energy and turbulent dissipation rate are performed in this study for different submergence depth, to find out which depth to be considered as minimum submergence depth for better efficiency and work of suction pipe and pump.
3. To investigate the issues occurred in the suction pipe like cavitation by studying pressure distribution for different submergence depth.

4. To examine the use of numerical software like computational fluid dynamics in the plan of suction pipe for sump pump.
5. To validate model, comparison of numerical simulation with the experiment conducted in Korea university[1] is done.

1.6 REPORT OUTLINE

In this report, the first chapter represents a brief introduction regarding the subject; the second chapter gives an assessment on the design of suction pipe for sump pumps for numerical modelling and measures to be taken to keep any unfavorable hydraulic conditions happening if any. The third chapter briefs about the methodology on the turbulent flow and suction pipe intake design as per HIS standards. The fourth chapter gives results for the simulation carried out in CFD. The fifth chapter gives synopsis, conclusion, measures and extent on future works.

CHAPTER-2

LITERATURE REVIEW

2.1 GENERAL

In this section, review of literature on design of suction pipe for better approach flow for pump efficiency is explained. Numerous analysts have conducted experiments on scaled models to study about the flow characteristics, hydraulic conditions influencing the performance of the pump and proposed the measures that are necessary to better the efficiency of the pump intake system. Use of numerical computational systems like CFD (Computational Fluid Dynamics) in the design of suction pipe for sump pumps is likewise studied. Thus, the outcomes on the experiments and numerical simulations, which have been done, is represented in this section.

2.2 VORTICES

A vortex is commonly related to the rotating movement of fluid around a common centreline. It is described via vorticity in the fluid, which measures at which rate the local fluid rotates. Typically, the fluid circulates around the vortex, the speed will increase as the flow approaches the vortex and the pressure decreases. Vortices can be found in nature and technology, which says vortices can be in a huge range of sizes according to the standards and risk management data which are as given below :

- superfluid vortices consist diameters around 10−8 cm (= 1˚A)
- trailing vortex of Boeing consist diameters around 727 1–2 m
- dust devils consist vortex of diameter 1–10 m
- tornadoes consist vortex of diameter 10–500 m
- hurricanes consist vortex of diameter 100–2000 km

The distribution of velocity, vorticity (the curl of the drift speed), as well as the circulation are used to study vortices. In maximum vortices, the fluid drift velocity is finest next to its axis and decreases in inverse percentage to the distance from the axis.In the absence of external forces, viscous friction inside the fluid tends to form irrotational vortices, in all likelihood superimposed to large-scale flows, such as larger-scale vortices. A shifting vortex contains with it a few angular and linear momentum, strength, and mass.

Formation of vortices is one of the major problem for better efficiency of a pump. Three types of vortices are normally found in the intake structures. They are

1. **Dimple Vortex**: A free surface vortex making a dimple on the free surface without air entrainment. Fig.1 shows the formation of a dimple vortex.
2. **Air Entraining Vortex**: A vortex which enters an intake from the free surface with intermittent or continuous air entrainment as shown in Fig.1.
3. **Submerged Vortex (Swirl):** A vortex which enters the intake from a solid flow boundary with submerged vapour core as shown in Fig.1

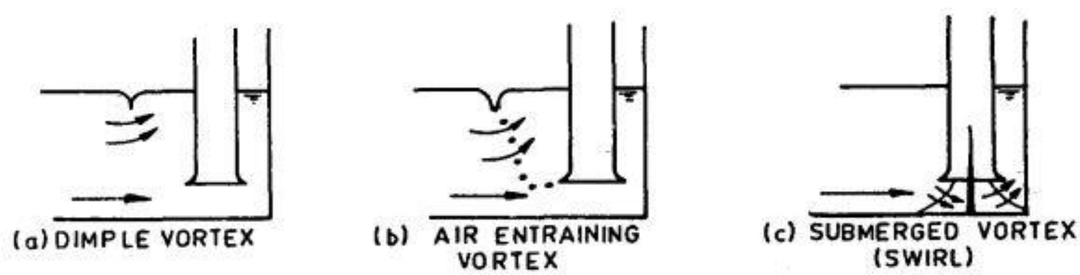


Fig. 1 : Vortex Classification

(Source: HIS 1998)

2.2.1 VORTICITY

Vorticity is a vector that describes the local rotary movement at a point in the fluid. Mathematically, the vorticity is defined as the curl (or rotational) of the velocity area of the fluid, normally expressed as

$\vec{\omega} = \Delta \times \overrightarrow{U}$

$\vec{\omega}$= vorticity vector

$\Delta \overrightarrow{U}$= local flow velocity

2.2.2 TYPES OF VORTEX ACCORDING TO FLOW

1. Rotational Flow
2. Irrotational Flow

1. ROTATIONAL FLOW

In this fluid velocity increases proportionally to the radial distance from the axis. If an element of fluid rotates about its centre as it was part of the body then the vorticity (ω), of such flow is same everywhere and its direction is parallel to the axis and its magnitude will be twice the angular velocity, Ω.

$\vec{\omega} = 2\vec{\Omega}$

$\vec{\omega}$= vorticity

Ω= angular velocity

2. IRROTATIONAL FLOW

Free vortices are irrotational vortices; they evolve quickly when fluid velocity is inversely proportional to radial distance. In free vortices, circulation is zero. In irrotational vortices, there is a core region surrounding the axis where the particle velocity stops increasing and then decrease to zero as radial distance goes to zero.

$\vec{\omega}$=0

To classify any flow as Rotational or Irrotational the angular movement of the fluid elements is analysed. If the angle among the two intersecting lines of the boundary of the fluid part changes at the same time as moving inside the drift, then the flow is Rotational Flow. But if the fluid element rotates as an entire and there may be no change in angles among the boundary lines then the flow be Irrotational Flow.

This means that there should be few deformation within the fluid element in a Rotational Flow. Such deformation of the fluid element or the shear strain is necessarily due to tangential forces or shear stresses. Shear stresses are as a result of viscosity, consequently the drift of viscous fluids is rotational. But this doesn't imply that the flow of non-viscous or ideal fluid is continually irrotational.

2.3 Background

Let D be a region in 3D space containing a fluid, and let $x = (x, y, z)^T$ be a point in D. The fluid motion is described by its velocity u(x, t) = u(x, t)i + v(x, t)j+w(x, t)k, and depends on the fluid density ρ(x, t), temperature T(x, t), gravitational field g and other external forces possibly acting on it. The fluid vorticity is defined by $\omega = \nabla\times u$. The vorticity measures the local fluid rotation about an axis, as can be seen by expanding the velocity near $x = x^0$,

$u(x) = u(x^0) + D(x^0)(x - x^0) + 1\ 2\ \omega(x^0) \times (x - x^0) + O(|x - x^0|$

where

$D(x0) = 1/2\ (\nabla u + \nabla u^T)$,

$$\nabla u = \begin{bmatrix} u_x & u_y & u_z \\ v_x & v_y & v_z \\ w_x & w_y & w_z \end{bmatrix}$$

The first term $u(x^0)$ corresponds to translation: all fluid particles move with constant velocity $u(x^0)$. The second term $D(x^0)(x - x^0)$ corresponds to a strain field in the three directions of the eigenvectors of the symmetric matrix D. If the eigenvalue corresponding to a given eigenvector is positive, the fluid is stretched in that direction, if it is negative, the fluid is compressed. Note that in incompressible flow $\nabla \cdot u = 0$, so the sum of the eigenvalues of D equals zero.

Thus at least one eigenvalue is positive and one negative. If the third eigenvalue is positive, fluid particles move towards sheets. If the third eigenvalue is negative, fluid particles move towards tubes. The last term in Eq. (1), $1/\ 2\omega(x^0) \times (x - x^0)$, corresponds to a rotation: near a point with $\omega(x^0) \neq 0$, the fluid rotates with angular velocity $|\omega|/2$ in a plane normal to the vorticity vector ω. Fluid for which $\omega = 0$ is said to be irrotational.

2.4 Minimum Submergence Depth

The basic design requirements of satisfactory hydraulic performance of rectangular intake structures include:

- Minimum depth of flow for uniform flow of the fluid in pump bays is required and it reduce the formation of vortices.
- Minimum pump bay width to depth requires limited inlet velocity of 0.5 m/s,to channel flow uniformly.

The minimum submergence S required to prevent strong air core vortices is based on dimensionless number, Froude's number as (from Bansal R.K)

$$F_r = \frac{v}{\sqrt{gD}} \tag{2.1}$$

Where,

F_r = Froude number

v = velocity in suction inlet

D = Outside diameter of belt or pipe inlet

g = acceleration due to gravity

The minimum submergence S can be calculated as

$$S = D\,(1 + 2.3F_r) \qquad (2.2)$$

The basic recommended layout for rectangular sumps, dimensioned in units of pump bell diameter "D" is as shown in fig.2

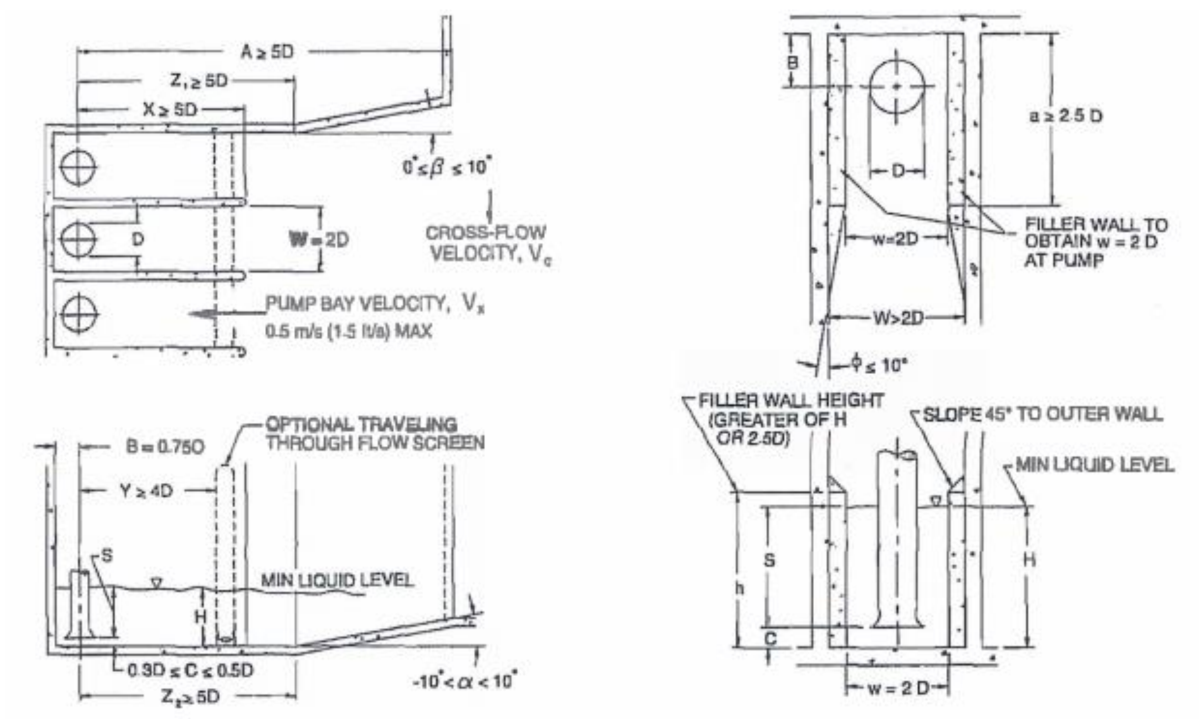


Fig. 2: Intake structure layout and Filler wall details with bay width

(Source HIS :1998)

These intake structure layout works efficiently according to the flow characteristics and on the geometry, boundary conditions, hydraulic conditions. Negative values of β (angle of wall divergence) require flow distribution or straightening devices, and should be developed with aid of a physical hydraulic model study.

Occasionally it is necessary to increase the bay width to prevent velocities at the entrance of pump bays from exceeding 0.5 m/s. Greater bay width may also result due to the arrangement of mechanical equipment. In these cases, the bay width in the immediate vicinity of the pumps must be decreased to 2D. For pumps with design flows of 315 l/s or less, no partition walls between pumps are required, and the minimum pump spacing shall be 2D.

2.4.1 Formed suction intakes:

The formed suction intake (FSI) may eliminate the need for the design of sumps with approach channels and appurtenances to provide satisfactory flow to a pump.

- Dimensions

The FSI design dimensions are indicated in Fig.3 .The wall shown in Fig.3 above the FSI opening reduces the tendency for surface vortices when the FSIs are installed in individual bays. The wall is not necessary for unrestricted approach flow conditions.

- Application standards

Minimum submergence is calculated as follows:

$$S/D = 1.0 + 2.3\, F_r \qquad (2.3)$$

Where:

S is the distance from the minimum recommended liquid level to the centreline of the FSI opening in the elevation view

D is the diameter of a circle having an area equivalent to the rectangular FSI opening,

$$D = [(4/\pi) W H_f]^{0.5} \qquad (2.4)$$

V used in F_r, is the average velocity through the FSI opening

W is the pump inlet bay entrance width

H_f is minimum liquid depth

2.5 STATUS OF RESEARCH WORK

Padmanabhan and Hecker (1984): conducted tests by building one full-sized and two reduced scale models of a pump sump to geometric sizes of 1:2 and 1:4 to decide if scale impacts the predictive capacity of hydraulic models of pump sumps with such substantial scale proportions. They had inspected the impacts of scale consequences for vortex generation, pipe swirl, inlet losses and air ingestion because of air drawing vortices. The outcomes demonstrated no huge consequences with the expectation of free surface vortex development in the test ranges. Pipe flow swirl and losses showed a few effects. The air-core vortices experienced were by and large powerless with low air withdrawals. Inside the precision of estimations of air-withdrawal rates, the model did not showed any huge prediction of air withdrawals.

Rajendran (1999): a simple pump sump modelled using appliances of CFD and validated their model with the results of data acquired from experimentation through PIV (Particle Image Velocimetry) estimations of physical model. The quantity of vortices formed, locations and their structures projected by numerical techniques were observed to be approximately same than those seen in experimentation. However, the vortices were observed to be for the most part bigger and weaker than what anticipated by the model.

Arboleda and Fade1 (1996): done model investigations to decide how the flow conditions in approach regions must be assessed to decide their effect on the performance of the pump sumps. The plan of sump pump was done in such way that it depends on approaching flow conditions. The study outlined the impact on water hydraulic performance and sump setup of several design elements and site obligations, which were necessary in the approach regions. The impacts of approach flow conditions on hydraulic performance were decreased with flow dispersion and structures like baffles, horizontal beams, ramps and flow splitters, which gave agreeable flow conditions. However, the flow distribution and guidance components can't be chosen theoretically and physical displaying is the main strategy to guarantee that.

Constantinescu and Patel (1998) : to simulate the three-dimensional flow field in a pump intake and to envisage the formation of free-surface and wall vortices, CFD model was used. The model solved the Reynolds-averaged Navier-Stokes equations and two turbulence-model equations in summed up curvilinear coordinates utilizing a completely implicit, fractional-step algorithm which is based on alternate direction implicit and approximate factorization. A model of two-layer k-∈ turbulence used determine wall flow that important to the interpretation of wall vortices. According to the plan criteria, very weak vortices were the outcomes. The predicted vortex structure was analysed.

Constantinescu and Patel (2000): to simulate the flow in a water pump intake bay,a numerical model was formed. The flow characteristics which they have considered are subsurface vortices in which they are attached to walls of the channel and free surface vortices which were seen in the research facility models. They were concerning the part of turbulence model and the impact of wall roughness in the expectation of location, size and strength of various vortices and the level of swirl in intake of pump and as needed calculations were done with k-∈ and k-ω models to anticipate vortices of same shape and size. The location and strength relied on upon the turbulence model and treatment of wall flow. Weaker vortices were expected when wall roughness was considered which raised the chances of using roughness as a device for vortex suppression.

Nagahara et al. (2003): experiments conducted by employing Particle Image Velocimetry (PIV) to estimate vortex flow. The mechanical assembly comprises of a suction intake and pressure tank, which can control the mean velocity for inlet, circulation and pressure at the bell mouth pipe. Vortices of diverse qualities produced and velocity fields measured and contrasted with the CFD outcomes. CFD modelling is finished using k-∈ turbulence method. The outcomes demonstrated that time averaged velocity profiles for the most maximum velocities are in agreement with the numerical ones. The maximum velocities acquired from instantaneous measurement outcomes are larger than time-averaged data and the core radii are significantly small to attribute to the unsteady movement of development of the vortex which demonstrated that CFD can’t precisely anticipate the detailed velocity profile in the region of the vortex centre and though when the computational grid is sufficiently fine to separate the small core area.

Hong-xun and Jia-hong (2007): model study conducted on a pump sump comprising of 5 pump intakes. They additionally completed numerical analysis for three-dimensional turbulent flow in the sump of the pump station. They proposed a boundary condition for the flow in the sump with several intakes at various flow rates. Finite volume method was used to solve the governing equations with the body fitted grid created by the multi-block grid system. Software fluent, was applied to compute the fluid flow in a model sump of the pump station. They discovered that numerical outcomes were genuinely good when contrasted with experimental results.

Tomoyoshi and Jun Matsui (2007): a model used to perform test for the performance of pump sumps, which are decreased in size to limit the construction costs .Experiments were done to measure the undesirable vortices, for example, air entrained and submerged which happened because of decrease in size .Critical submergence for flow rates was inspected through visual perceptions through a video recorder. Velocity and vorticity distribution in the sump were measured with PIV instrument. Studies were done to limit the undesirable vortices like air entrained and submerged vortices. CFD studies were conducted and they contributed codes, for example, "Virtual fluid system 3D","Star CD-3.22","Star CD-3.26" and "ANSYS CFX-10.0".The computed outcomes were contrasted with the experimental results for flow designs, location of vortices and vorticity.

Hai-feng and Hong-xun (2009): to research the formation and change in development of the free surface vortex an experimental model was set up. A Particle Image Velocimetry (PIV) measured the free surface vortex flow field at various stages. Flow visualization study was done to find the vortex position and discover its structure. Empirical formulas for the critical submergence and the entire field structure were found. It is discovered that the tangential velocity distribution is not much different from the Rankine vortex and the radial velocity changes little in the vortex zone. Vortex begins from the free surface and progressively escalates to air entrainment vortex. The vortex center moves amid the development and advancement of the free surface vortex. In accordance with experiments, the vortex position and structure were found by numerical simulation joined with a vortex model and contrasted with the results of experiments, which indicates acceptable results.

Choi et al. (2010): model tests were conducted to study the flow design around the pipe intake structure. The flow consistency in the intake channel was inspected to discover the reason for the cause of vortices. A multi-intake sump model consist of 7 pump intakes and a single intake sump model considered for examination. Anti-submerged Vortex Device(AVD) were introduced in the single and multi-intake pump sump models and their adequacy in diminishing the vortices was observed both by experimental and numerical techniques. The outcomes demonstrated that most high estimation of flow uniformity is at the inlet of pump intakes 3 and 5 in the multi intake pump sump with 7 pump intakes. So amid the plan of intake system the flow design at the upstream zone of pump inlet in the fore bay should be considered in detail to keep the vortex arrangement around the bell mouth. Strong submerged vortex effectively decreased by AVD in the intake just underneath the bell mouth.

Desmukh and Gahlot (2010): a study was conducted in order to check the feasibility of a software CFD as advancement tool for pump sumps. In their investigation commercially accessible programming software ANSYS CFX has been utilized for CFD examination of flow conditions in a pump sump and the outcomes got were observed to be in good condition with the experimentally obtained results. A pump sump comprise of a main channel, approach channel, fore bay, pump sump and intake with a scale proportion of 1:11 was utilized for experiments. The time and cost required in sump study demonstrated geometry and design for sump optimization can be reduced due to CFD studies.

Lucino and Gonzalo Duro (2010): led investigations to decide the feasibility of the computational fluid dynamics (CFD) in the study of vortices in the pump sump. Determined velocity corresponded well to patterns and magnitude of measured ones, while the estimations of maximum values for vorticity computed were higher than those measured, which was clarified by the characteristics of estimation in the physical model. Model anticipated the occurrence of surface vortices which were found in the physical model, some of the wall vortices and floor vortices which were not found in the model were also found. For low submergences, the floor vortex in the physical model is gathered from the cavitation in the centre. The portrayal of total vorticity transformed into an extremely helpful tool to picturize and consider the actual location, direction and time difference of concentrated vortices.

Shazy and Shabayek (2010): directed analyses on a bodily hydraulic model of a circulating water pump sump shape. The cooling water created from circulating water pumps and auxiliary water pumps pulling again go with the flow out of one aspect of cooling tower basin. In the primary research intake structure forms high stage of pre swirl and sturdy vortices on the intakes of pumps. Therefore, they've made few changes in the geometry to decrease the pre swirl and vortex development by using introducing sidewall fillets; back wall fillets and ground splitters. They also added curtain wall set at El. 1089.95m and three.71m from the returned-wall of the sump. With these changes, they watched go with the flow pre swirl, vortices and throat velocities inside the pump sump to be in quality limits operating situations for model. The advocated minimal water degree within the cooling tower basin to keep association of a hydraulic jump and to keep away from unfavorable hydraulics and degradation in pump execution inside the sump became El. 1089.90om.

Pradeep et al. (2011): built a model set-up with a supply tank sustaining to the intake system through an inlet taken after by a short approach channel, a slope transition, expanding fore bay and sump chamber with a scale proportion of 1:11. Numerical simulation (CFD) examines were completed utilizing FLUENT programming which included estimation of factors, for example, velocity distribution at the cross area of draft tube inlet, swirl angle in the pump suction pipe and streamline designs in the sump chamber. Four unique cases with three distinct sets of discharges and three diverse working conditions characterized by number of pumps running both symmetrically and asymmetrically were tested. Numerically calculated average velocities,vertical velocity profiles at the draft tube channel and swirl angle at the pump suction pipe were observed to be in good agreement with the experiment ones. CFD likewise anticipated the formation of a circular zone in the fore bay, which was in good agreement with the experimental ones in spite of the fact that its size was more than that assessed by CFD.

Jie-min etal (2012): utilized a 3D numerical model for the pump intake model considered Navier Stokes equation with RNG k-∈ turbulence model and VOF technique to simulate the free surface. The experiments were carried out on a non-symmetric pump intake model to compare the outcomes and decide the relevance of the model. Five different system of intakes comprising of three lateral and two front intake systems were taken to observe the flow designs at intakes and their consequences for pump operation. They discovered that asymmetry in the intake structure prompts the advancement of the vortex formation bringing about undesirable flow designs and hence vital changes were made in the structure to diminish the inadequacies. The anticipated areas, structures and shapes of all vortices were observed to be in good agreement with those seen in experiments having different vorticity strengths. The flow design and the efficiency of five other pump intake system were studied. The discharge and the velocity consistency of the intake system were used to study the performance.

Meena et al. (2013): directed physical model investigations in the lab to examine the performance of pump sumps in cooling water system for thermal power plants. Through scaled physical model investigation in research center, the proposed configuration is checked and considered for alterations to the intake geometry can be recommended. On the other hand, physical model investigation is very complex and costly. Numerical model of the intake structure utilizing CFD (Computational Fluid Dynamics) programming can be completed to get appropriate sump intake plan. In this paper, numerical examination of a pump sump model is done by utilizing ANSYS FLUENT programming software. The hydraulic model investigations of a specific intake sump were done in the Hydraulic Engineering Laboratory of the Civil Engineering Department of Indian Institute of Technology Bombay and comparing CFD examination is done utilizing FLUENT. Both outcomes are compared and it is discovered that the simulation outcomes are approximately same with the experimental results. The investigation additionally helped in understanding the part of numerical simulation in saving model cost and sparing valuable time.

Pratap and Chavan (2013): directed investigations to model flow characteristics in a pump sump of physical model by using Computational Fluid Dynamics (CFD) code FLUENT. The experimental methods considered collecting data by flowmeter and swirl meter (Rotometer/Vortimeter). Two sorts of estimations were done which are flow; and swirl angle. A visual test that includes the dye tracing method was conducted to study flow characteristics. The CFD investigation is done at critical cases, grid generation is done in ICEM-CFD and numerical analysis are done in FLUENT, and flow is studied with the assistance of velocity stream lines and vector plot and velocity contour at intake of pump sump, in CFD-POST software and compared with experimental and CFD result. Subsequently this work is considered for deciding the feasibility of CFD software as a design optimization tool.

Kim et al. (2015) : directed examinations to check the flow conditions around the intake of suction pipe . A sump model was built with a scale proportion of 1:10.Experiments were conducted and the flow conditions around the suction pipe were considered. Uniformity of flow tested in the intake channel to discover the causes for vortex development. Anti vortex design were introduced in the single pump intake model to smoothen the vortex development and their adequacy is examined. Numerical simulation is done with SST turbulence method for the expectation of location of vortex development. CFD and experimental studies were done with and without AVD's and they yielded same outcomes. Without the AVDs, the maximum swirl angle acquired for experimental and CFD investigation were 10.9 and 11.3 degree separately. Correspondingly, with AVDs, the maximum swirl angle acquired for experimental and CFD examination were 2.7 and 0.2 degree separately. As indicated by the ANSI/HI 98 standard that allows a maximum swirl angle of 5 degree, the utilization of AVDs in experimental and CFD examination created exceptional outcomes that were well under the limit.

2.6 CLOSURE

Literature review establishes that we can use experimental or numerical strategies in the design of suction pipe for betterment of pump sump system. These studies were conducted keeping in mind to perfect the imperfect design and performance in the pump geometry which leads the decline in the efficiency of the pump. By directing model studies different undesirable phenomena like swirl, vorticity and so on can be resolved and proper measures can be taken to anticipate them. Numerous analysts utilized softwares like ANSYS FLUENT to simulate the flow conditions using Reynold's Averaged Navier Stokes equation (RANS) and different flow phenomena like velocity, vorticity at various locations are acquired. A contrast is done between both the results and they are observed to be close. As the numerical simulation results are observed to be in great concurrence with the experimental ones they have an edge so as to overcome the barrier of cost and time. Some different works are done to decide the position of suction pipe of sump so that the unsettling influences are limited. Hence model studies plays a vital role in design.

CHAPTER-3

METHODOLOGY

3.1 INTRODUCTION

Numerical models have the potential to atleast partially replace the costly physical model tests. Numerical models not only save the cost of various expensive laboratory instruments but also time spent for the conductance of the experiment. Even though the numerical simulation cannot replace the experiments completely, the amount of experimentation and the overall cost can be significantly reduced. The major problem with scale-down laboratory tests is that they have scaled effects. The numerical models don't undergo from scale effects.

Once a numerical model is developed, it can be subjected to various boundary conditions. We can even apply those conditions which are difficult to impose on laboratory models. It is well known that a well-designed numerical model can certainly be complementary to model tests and can assist researchers in identifying the most important cases for which the model tests were conducted. The results of a simulation can never be totally reliable due to the inadequacy of mathematical model. Due to the limitation of available computing power the reliability of simulations is affected. The use of Computational fluid dynamics software is very much helpful in solving partial differential equations based on conservation principles. Some of the software used were FLUENT, PHOENIX, Open FOAM, CFX etc. CFD is not only used by hydraulic engineers but also chemical engineers, petroleum engineers and military organizations, mechanical engineers, aerospace engineers and so on.

3.2 COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics (CFD) is the qualitative and quantitative prediction of fluid flows. It can also predict the heat and mass transfer, compressible and external compressible flow, convection, radiation, fluid mixing in porous media etc. CFD uses numerical methods containing a set of governing equations and algorithms to solve and analyse problems that involve such fluid flows. It is essential that designers or analysts should have a basic knowledge of the underlying concepts involved. This is the reason why validation which involves checking if the model itself is adequate for practical purposes is done. The goal of validation is to ensure that the results produced by CFD are reasonable. CFD analysis compliments testing and experimentation by reducing total effort and cost required for experimentation and data acquisition.

3.3 MATHEMATICAL FORMULATION IN CFD

In the mathematical formulation of CFD, a set of governing equations were numerically solved which consists of conservation equations for mass and momentum. There are some additional transport equations which are required to be solved for turbulent flows.

For the CFD analyses, following governing equations are used. (ANSYS 2016)

1. Mass Conservation Equation

$$\frac{\partial p}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j)=0 \qquad (3.1)$$

2. Momentum Conservation equation

$$\frac{\partial}{\partial t}(\rho u_j)+\frac{\partial}{\partial xj}(\rho u_j u_i + p\delta_{ij} - \tau_{ij}) + \rho g^{\rightarrow}+F^{\rightarrow} = 0 \qquad (3.2)$$

Where and gravitational and external body forces respectively.

3 .Energy conservation equation

$$\frac{\partial e}{\partial t} + \frac{\partial}{\partial x_j}(\mathrm{u}\ (\mathrm{e+p}) - \mathrm{u}\tau_{ij} - q_j)=0 \qquad (3.3)$$

Where ρ is fluid density, u is flow velocity vector field, p is pressure,τ_{ij} is normal stress, e is energy, t is time, $\delta_{ij}$is Kronecker symbol (δ_{ij}= 1 when $i = j$,$\delta_{ij} = 0$ when $i \neq j$), q is heat flux.

Other equations used in CFD are,

4. conservation of species

5. effect of body force

3.4 ANSYS FLUENT

ANSYS Fluent software contains the wide physical modelling abilities required to model flow, turbulence, chemical reaction and heat transfer for mechanical applications and more. ANSYS Fluent software is an important part of the planning, optimization and design parts of model development. ANSYS Fluent works on the principle of Finite Volume Method (FVM) where the domain are splitted into finite set of control volumes.

The finite volume discretization is based on an integral form of the partial differential equation (PDE) to be solved. The PDE is written in a form of a given finite volume (or cell). The computational domain is divided into finite volumes and then governing equations are solved for each cell. The main advantage of FVM over FDM is that, it doesn't require the use of structured grids and the effort of converting the given mesh into structured one is completely avoided. In FVM values of field variables at non- storage locations such as vertices are also obtained by interpolation. Advanced solver technology provides fast, accurate CFD results, flexible, moving and deforming meshes, and superior parallel scalability

3.5 FLUENT CFD SIMULATION

Fluent basically divide the total structure of domain into small volumes or cells. As the number of cells increases, the calculations will be accurate and it will take more time to execute. For each problem, the entire CFD modelling involves 4 steps.

1. Problem identification,

2. Pre-processing,

3. Solver execution

4. Post processing.

The numerical simulation chart is given below in fig.3 :

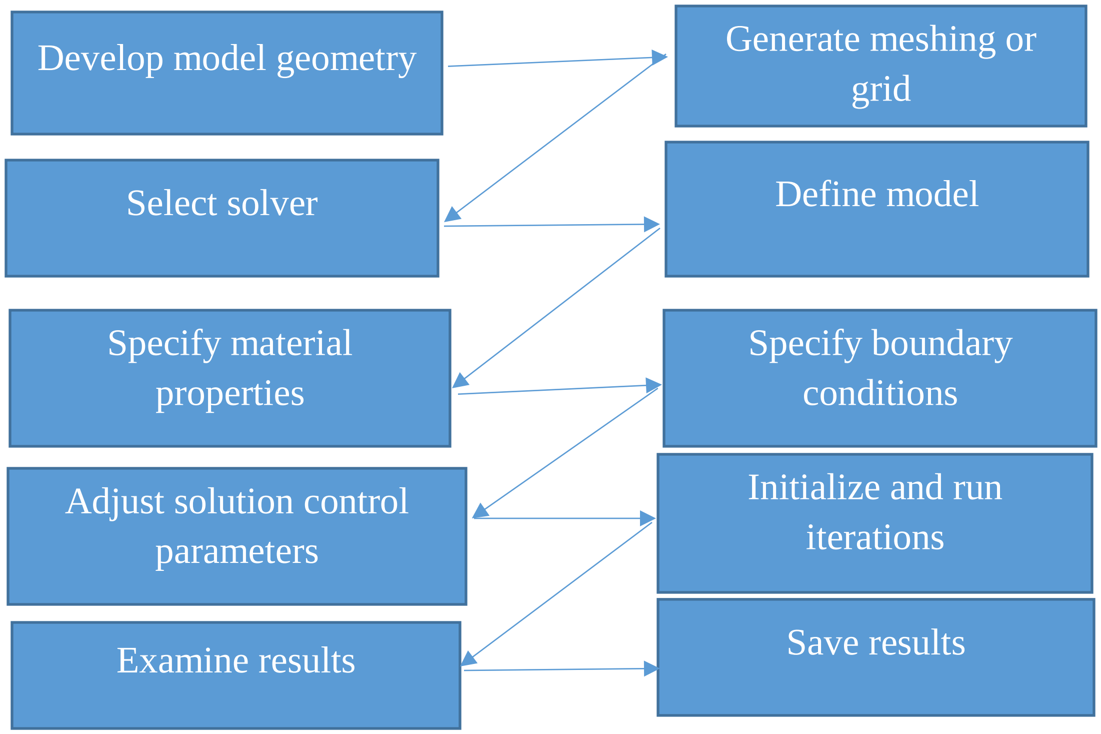


Fig. 3: Steps involved in Fluent Simulation

(Source: ANSYS Fluent Guide)

3.5.1 Geometry

The Design Modeller application within the FLUENT can be used to create new model geometry. The Design Modeller application can be used to create 2D sketches, modelling 3D parts, or directly importing 3D drawing models for pre-processing. It provides with a function to convert to 2D sketches into 3D models. It also consists of various geometric shapes with which allows adding or cut material from a model. Similar to Design Modeller, other software such as Gambit, ANSYS ICEM, Solid Woks etc can also be used which will be more user friendly than that of design modeller. All the features are available as that of design modeller interface but importing the geometry may be bit difficult. In our case, a rectangular sump of length 1.29m, depth as water depth and width as 0.5m is to be taken. Then the pier section is drawn at the desired point with the dimensions for the water depth.

3.5.2 Mesh Generation

To analyse fluid flows, flow domains are split into smaller sub domains or elements because the fluid flow governing partial differential equations are not generally responsive to analytical solutions. Inside each of these sub domains governing equations are discretized and solved. Commonly three methods are used to solve the approximate version of the system of equations: Finite Volume Method, Finite Element Method, or Finite Difference Method. While meshing, there must be ensured that there is proper continuity of solution between the common interfaces between two sub domains. The gathering of all elements or cells is known as mesh or grid. The Automatic Method of meshing needed to consider between Tetrahedral Patch Conforming and Swept Meshing, which depends on the body if it is sweepable or not. The quality of meshing is important as the accuracy of the results depends on it to a large extent. A good mesh minimizes the errors in the solvers leading to numerical diffusion. A good mesh has three components:

- Good resolution
- Appropriate mesh distribution
- Good mesh quality

3.5.3 Solvers Used

The solver used is pressure based, steady and absolute velocity transformation. Here gravity in – Z direction is also taken into consideration. Standard K-epsilon turbulence model is used to simulate the fluid flow around the bridge piers in the flume. As the solvers are improved, rate of convergence of the simulation and the accuracy of the computed result will be increased. The turbulence models which are available in Fluent are as follows: (ANSYS 2013)

1. Standard, RNG, and Realizable k- ε Models.
2. Standard and SST k- ω Models
3. k- kl- ω Transition Model
4. Reynolds Stress Model (RSM)
5. Scale-Adaptive Simulation (SAS) Model
6. Detached Eddy Simulation (DES)
7. Large Eddy Simulation (LES) Model

The working of a solver selected will mainly depends on the choice of selection, discretization scheme, convergence criteria, accuracy. below the basic outflow of a simulation:

1. Set the solution parameters
2. Initialize the solution
3. Enable the solution iteration and monitors of interest
4. Calculate a solution
5. Check for convergence
6. If convergence occur, check for accuracy and go to results.
7. If convergence didn’t occur, modify solution parameters or grid.

Mainly there are two kinds of solvers available in FLUENT as shown in

– Pressure based

–Density based

The pressure-based solvers solves pressure and momentum as primary variables. Pressure-velocity coupling algorithms derived from the continuity equation by reforming it. Two algorithms available in relative to pressure-based solvers:

1. Segregated solver – solves momentum sequentially and pressure correction

2. Coupled Solver (PBCS) – Solves momentum simultaneously and pressure

3.5.4 Standard K-epsilon Solver

The standard k-epsilon model in ANSYS Fluent has become the workhorse of practical engineering flow. It is due to its robustness, economy, and reasonable accuracy for a wide range of turbulent flows, its gaining popularity. A semi-empirical model equations relies on empiricism and phenomenon logical considerations.

The standard k-ε model is a model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). The model transport equation for ε obtained using physical reasoning which is not as exact as its mathematical, while the model transport equation for k derived from the exact equation.

3.5.5 Standard model transport equations

The Turbulent kinetic energy (k) and its rate of dissipation (ε) are obtained from the following equation.

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left\{\mu+\frac{\mu_t}{\rho_k}\right\}\frac{\rho k}{\rho x_j}\right]+p_k+p_b-\rho\epsilon-Y_M+S_k \qquad (3.4)$$

For dissipation

$$\frac{\partial}{\partial t}(\rho\epsilon)+\frac{\partial}{\partial x_i}(\rho\epsilon u_i)=\frac{\partial}{\partial x_j}\left[\left\{\mu+\frac{\mu_t}{\rho_{k\in}}\right\}\frac{\rho\in}{\rho x_j}\right]+C_{1\in}\frac{\in}{k}(P_k+C_{3\in}P_b)-C_{2\in}\rho\frac{\in^2}{k}+S_\in \qquad (3.5)$$

Modelling turbulent viscosity, turbulent viscosity is modelled as

$$\mu_t=\rho C_\mu\frac{k^2}{\in} \qquad (3.6)$$

For dissipation of k

$$P_k=-\rho u_i' u_j'\frac{\partial u_j}{\partial x_i}P \qquad (3.7)$$

$$P=\mu_t k S^2 \qquad (3.8)$$

Where S is the modulus of the mean rate-of-strain tensor which is defined as:

$$S=\sqrt{2S_{ij}S_{ij}} \qquad (3.9)$$

u_i , u_j – velocity vectors

x_i , x_j – position vectors

k – turbulent kinetic energy

μ – kinematic turbulent viscosity

μ_t – turbulent eddy viscosity

S_{ij} – mean rate of strain tensor

$\in$ - turbulent dissipation

In the above equations, P_k represent the generation of turbulence kinetic energy due to the mean velocity gradients, P_b represent the generation of turbulence kinetic energy due to buoyancy, Y_M represents the contribution of the fluctuating dilation in compressible turbulence to the overall dissipation rates, $C_{1\in}$, $C_{2\in}$, and $C_{3\in}$ are constants. σ_k and σ_ϵ are the turbulent prandtl numbers for k and Є respectively. S_k and S_ϵ are user defined source term.

3.6 NUMERICAL SIMULATION

In this section CFD results obtained from Ansys Fluent are presented. In this study numerical analysis is done for a static case. That is, the simulation is carried out under steady state conditions. The aim of the analysis is to understand the flow behaviour. Boundary conditions are given as close as possible to get the simulated results near to the prototype. The closeness of the results depends upon the meshing, material property, cell zone and boundary conditions and local conditions which are prevailing. Simulations are carried out for various operating conditions of the pumps at varying depth of 50mm, 130mm and 300mm.

Flow domain of the model is set up and the results are simulated in the software and velocities, turbulent kinetic energy and turbulent dissipation are. There could be some errors, because of various local factors in the prototype run, which cannot be included in the software, due to the lack of options as the software packages are considering many ideal boundary conditions which may differ from actual conditions.

The geometry of the flow domain and meshing is done in Ansys workbench and post processing of the results is carried out in CFD post. Simulation carried out at three different depth of 50 mm, 130mm and 300mm conditions.

3.6.1 Model Setup

Model is constructed as shown in the fig. 4. Model is constructed with a scale of 1:12 for comparison of CFD models and 1:10 to study occurrence of vortices and characteristics of vortices around the suction pipe for change in submergence depth. The geometry of the model starts with an inlet to the sump followed by an approach section which is the rectangular part of the sump consisting of 3 pump bays. Pumps are placed in these bays consisting of bell mouth to enable smooth entry of flow into the pumps. Water is lifted into these pumps through siphon mechanism. The depth of the model is set to be 0.5m.

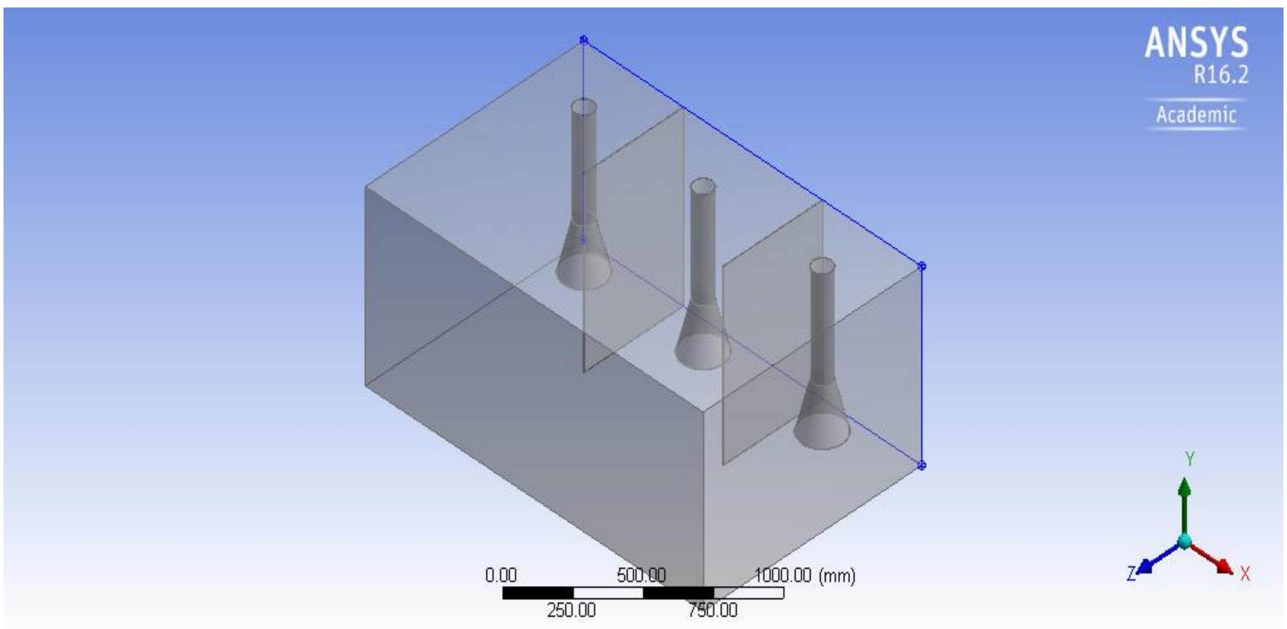


Fig. 4:3D view of the model

Computational investigations are carried out in ANSYS FLUENT. Meshing is done in Workbench and the simulations are carried out. In this model structured mesh with hex-dominant mesh is used for grid generation. The mesh consists of 456237 elements with number of nodes as 84897 for model 1:10 scale and for 1: 12. Special tools like body sizing, refinement are used in order to get a finer mesh. Different sections of the flow domain are named in order to distinguish the property of section.

Table -1: Model parameters of sump and suction pipe system

Parameter	Prototype	1:12 Scaled model	1:10 scaled model
Inlet velocity	1.203 m/s	$v_m = V_p/(12)^{0.5}$=0.347 m/s	0.381m/s
Discharge of each pump	q = 18000 Cum/hr	q = $q/12^{2.5}$ = 10.02LPS	56.9 m^3/hr
Maximum water depth in sump from bottom level of sump	D_{max} = 8m	d_{max} = $D_{max}/12$ = 0.67m	0.8m
Normal water depth in sump from bottom level of the pump	D_{normal} = 7m	d_{normal} = $D_{normal}/12$ = 0.583m	0.7m
Minimum water depth in the sump from bottom level of the sump	$D_{minimum}$ = 4m	d_{min} = $D_{minimum}/12$ = 0.33m	0.4m
Sump width	B = 34 m	b = $B/12$ = 0.333m	3.4m
Sump length	L = 65.8m	l = $L/12$ = 1.29167m	6.58m
Diameter of bell throat pipe	0.09m	0.0075m	0.009m
Diameter of bell mouth	0.2m	0.0167m	0.02m
Bell mouth ground clearance	1.3m	0.1083m	0.13m

Fig. 5 shows the top view of the model. Table2 gives the boundary conditions:

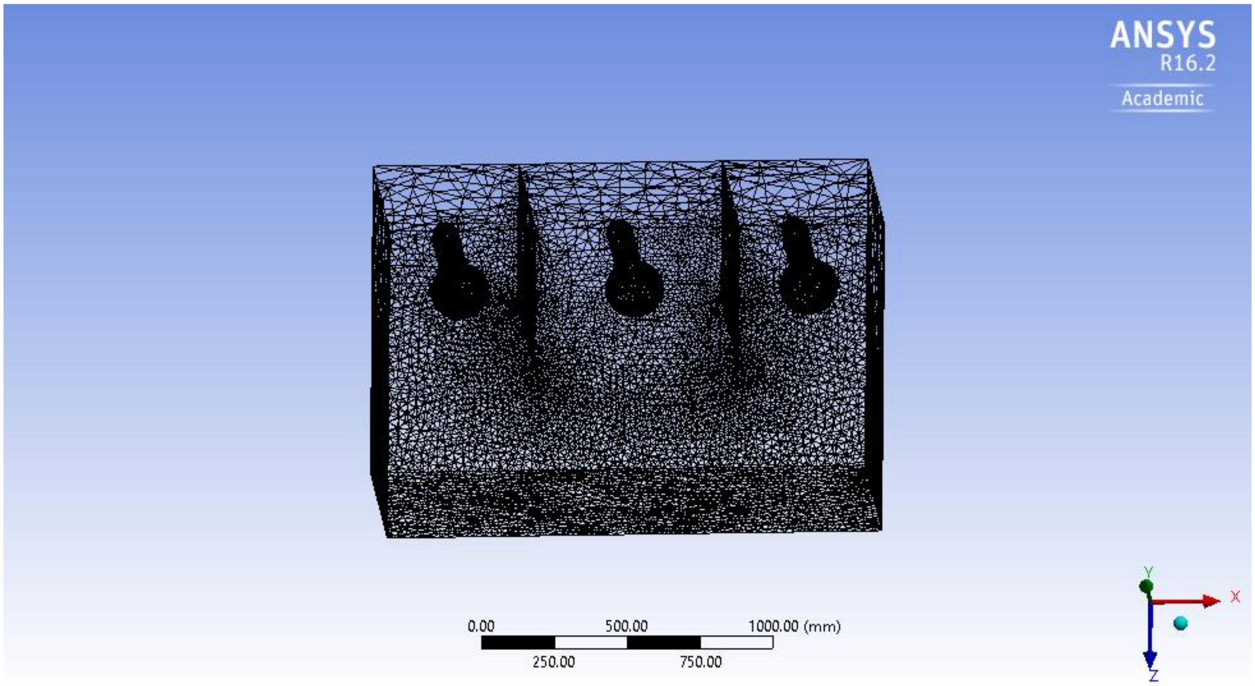


Fig. 5: Top view of the mesh domain

Table 2 Boundary Conditions at Different Phases

SECTION	BOUNDARY CONDITION
Inlet	Mass flow inlet = 36.08 m^3/hr Hydraulic diameter = 0.2 m Velocity inlet=0.347 m/sec
Outlet	Outflow
Side walls	Stationary wall No slip condition

	Roughness height – 0 Roughness constant – 0.5
Front walls	Stationary wall No slip condition Roughness height – 0 Roughness constant – 0.5
Inner sump walls	Stationary wall No slip condition Roughness height – 0 Roughness constant – 0.5
Top surface	Stationary wall No slip condition Roughness height – 0 Roughness constant – 0.5

The domain is specified as a Non Buoyant, stationary fluid with working fluid as water at 25℃ and reference pressure at 1 atm, The turbulence model is selected as standard $k{-}\in$ model. In the steady state the boundary condition at the inlet is specified as velocity inlet with a flow of 0.347 m/s for all the submergence depth condition. The pipe outlets are specified as outflow. The hydraulic diameter of inlet and pipe are 0.2m and 0.09m respectively. No slip shear condition is used for all the walls. For the shear condition of free surface the value of specified shear is taken as zero.

The model is then simulated using CFD solver, ANSYS. Solver is specified as pressure based and there is a gravitational force of -9.8m/s^2 in z direction. In Solution methods, simple scheme is adopted for Pressure Velocity Coupling. First order upwind is selected for momentum, turbulent kinetic energy and specific dissipation rate. Surface monitors are applied at inlet and pipe outlet.

Standard Initialization is used as an Initialization Method. Table 3 shows the boundary conditions used for simulation.

Table 3 : conditions for simulation

S.No	Settings	Choice
1	Simulation	3D
2	Solver	Pressure based
3	Model	Viscous standard $k - \in$ model,standard fun
4	Material	water
5	Pressure velocity coupling scheme	simple
6	Gradient	Least square cell based
7	Discretization pressure	Standard
8	Discretization momentum	Second order momentum
9	Turbulent energy dissipation	First order upwind
10	Under relaxion factors	Pressure – 0.3 Density – 1 Body forces-1 Momentum – 0.7 Turbulent kinetic energy- 0.8
11	Compute from	Inlet

3.7 CLOSURE

In this chapter, the model set up, how scaling of prototype is done and the geometry of models is discussed. Also the tools of CFD which were used to determine the flow rate, velocity and vortices are presented.

CHAPTER -4

RESULTS AND DISCUSSION

4.1 Simulation results

4.1.1 VALIDATION OF THE MODEL

The data of the experiments done in Korea university[1] is further used to validate the CFD obtained results. In the experimental result, location of vortex and swirl angle are determined via visual studies and using of vortimeter. The experiment was performed in Korea Maritime and Ocean University (KMOU) and the flow conditions around suction pipe structure were investigated. In this study, uniformity of flow distribution in the suction pipe examined to find out the specific causes of vortex occurrence.

1. SWIRL ANGLE**:**

Swirl angle calculates the intensity of flow rotation. HI standard is used to calculate swirl angle in experiments that also checks the flow rotation of suction pipe. Rotation time for swirl meter is above 10minutes used for observation. According to HI limit for swirl angle is 5º. Equation given below is to calculate swirl meter in experiments.

$$\theta = \tan^{-1}\frac{V_\theta}{V_Z} \tag{4.1}$$

Where

θ = swirl angle

V_θ = circumferential velocity

V_Z = axial mean velocity

To calculate swirl angle using CFD, average of tangential velocity is necessary. Below is the expression to find swirl angle:

$$V_\theta = \frac{2}{3} * \sum_0^N {V_{0.25D}}/{N} + \sum_0^N {V_{0.5D}}/{N} + \sum_0^N {V_{0.75D}}/{N} \quad (4.2)$$

In the following chart and table we can see that swirl angle is decreasing with the increase in depth. For the model there is no particular submergence depth obtained to minimize the swirl angle , hence to reduce we can use anti-vortex device, baffles, piers etc.

Table 4: Comparison of Vortex Region and Swirl Angle For Experimental and CFD Model for 130mm depth (Experimental Model Considered 130mm Depth)

For 130mm	Swirl angle ^{0}(degree)			LOCATION OF VORTEX REGION		
	PIPE 1	PIPE 2	PIPE 3	PIPE 1	PIPE 2	PIPE 3
EXPERIMENTAL	10.7^0	7.6^0	N/A	RSW	LSW	LSW
				BW	BW	BW
				B	B	B
CFD	10^0	6^0	11^0	LSW	LSW	LSW
				RSW	RSW	RSW
				B	B	BW

a-Right Side Wall, b-Back Wall,c-Bottom Wall,d-Left Side Wall

2. To Validate Location of Vortex Region Simulation Model is Shown Below**:**

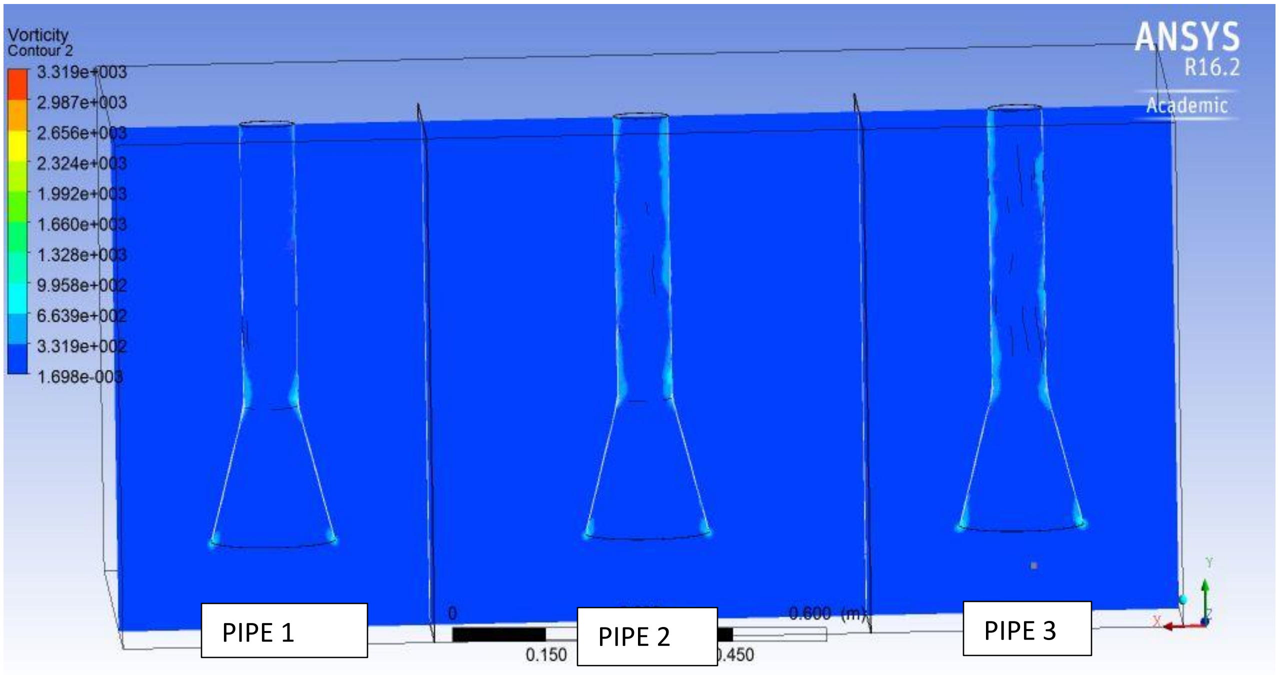


Fig. 6: Vortex Region obtained from CFD

4.1.2 COMPARISON OF K-Ɛ MODELS AND SHEAR STRESS TRANSPORT MODEL:

1. Vorticity and swirling strength

The formation of vertical vortex for model are simulated by Reliable k- Ɛ, standard k- Ɛ and SST model. In Fig.6 a superficial vortex formation occurs at t=0.9s in standard k-Ɛ model, and in Fig.7 a superficial vortex formation occurs at t=0.3s in shear stress transport model. In Fig.8 Reliable k-Ɛ model represent vortex formation at t=0.4s. The table 6.0-1 shows that vortex generates at different time step but the better model generated it earlier in time for approximately same swirling strength and vorticity. It represents that k-Ɛ model is not suitable than SST model for simulating vortices is, but standard k- Ɛ model is better than reliable k- Ɛ model. Vorticity describes the magnitude and direction of vortex where swirling strength tells about the intensity of vortices.

Table 5: accumulated time step

model	Time(sec)	Vorticity(s^{-1})	Swirling strength (s^{-1})
Standard k-ε	T=0.3s	216.345	86.55
SST	T=0.2s	220.981	84.874
Reliable k- ε	T=0.4s	224.561	89.136

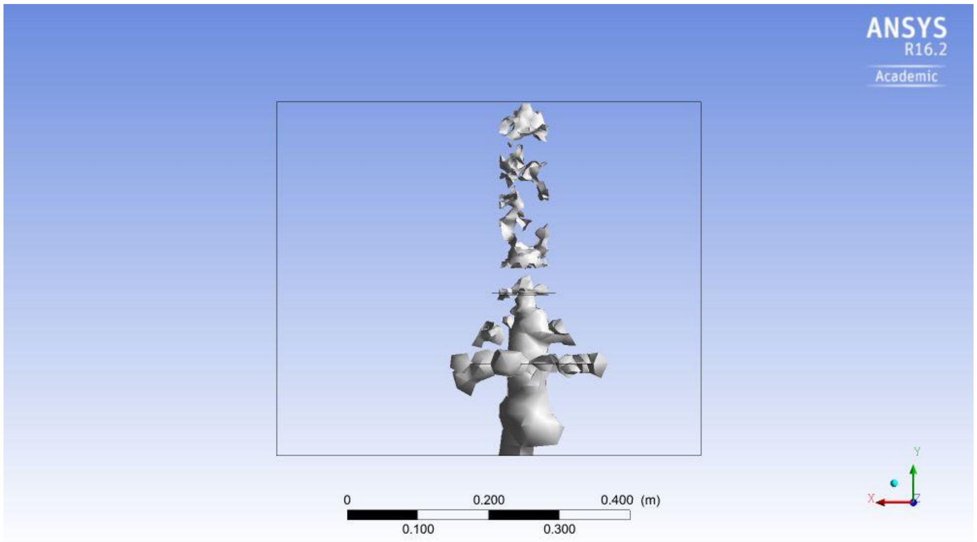


Fig. 7: standard k-Ɛ model (t=0.3s)

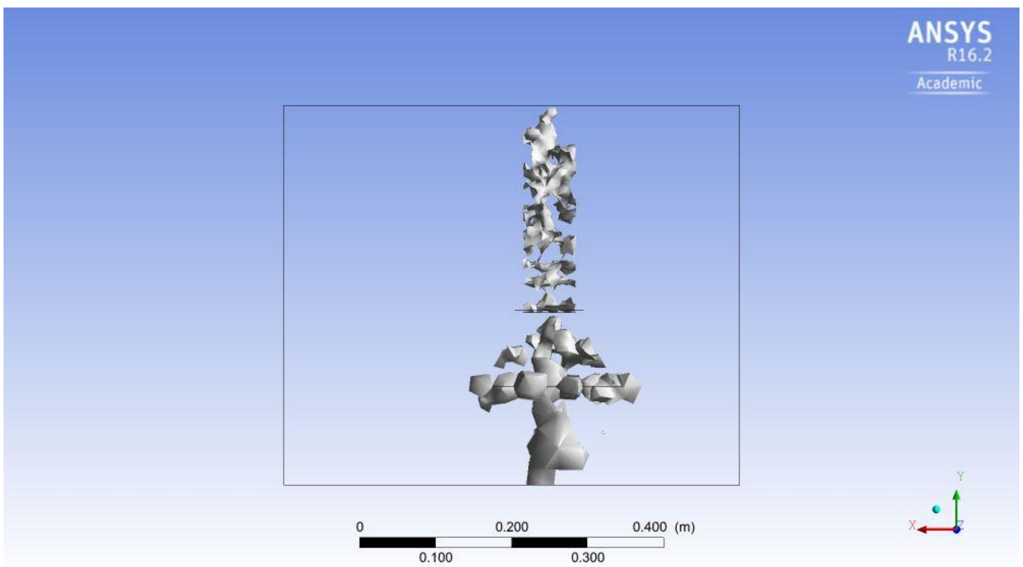


Fig. 8: shear stress transport model (t=0.2s)

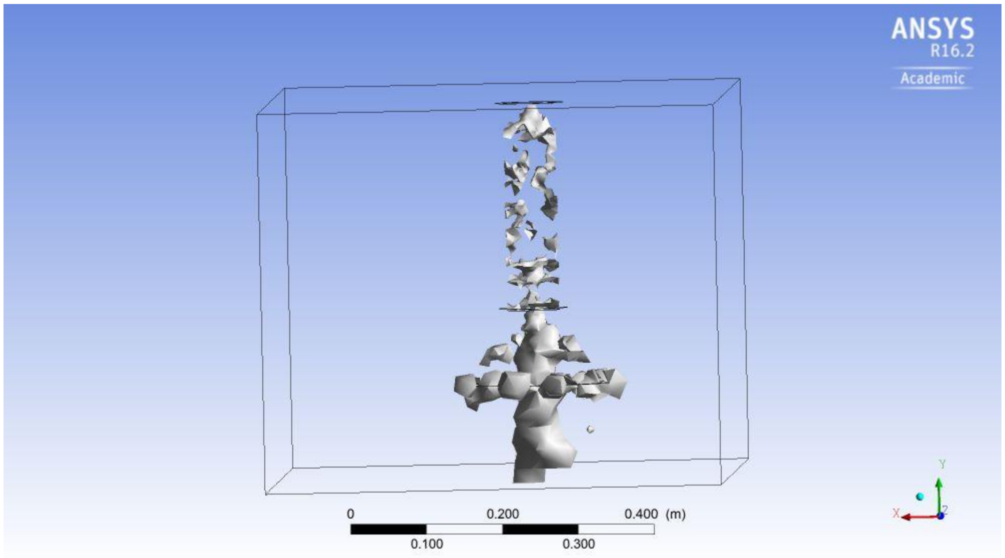


Fig. 9: Reliable k-Ɛ model (t=0.4s)

2. Volume fraction of water

Volume fraction of water shows the approach of flow to the suction pipe. The place where surface depression occur causes vortex core to originate, so this why condition of water surface is a key to know the vortex region. Following is the figures to show the difference of the model for volume fraction of water. In the following figures, it can be seen that shear stress transport model simulation is clearer than the other two model. Although SST model used for complex problem and one can rely on standard k-Ɛ model.

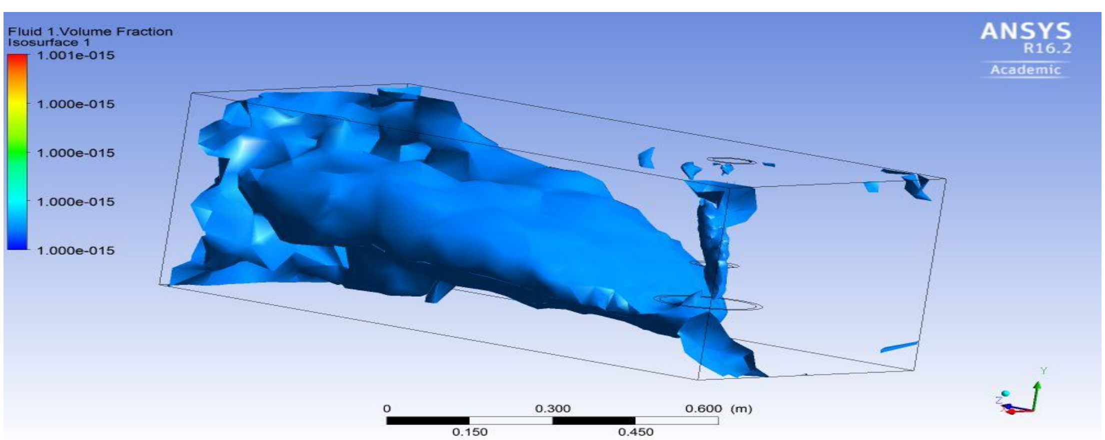


Fig. 10: SST model

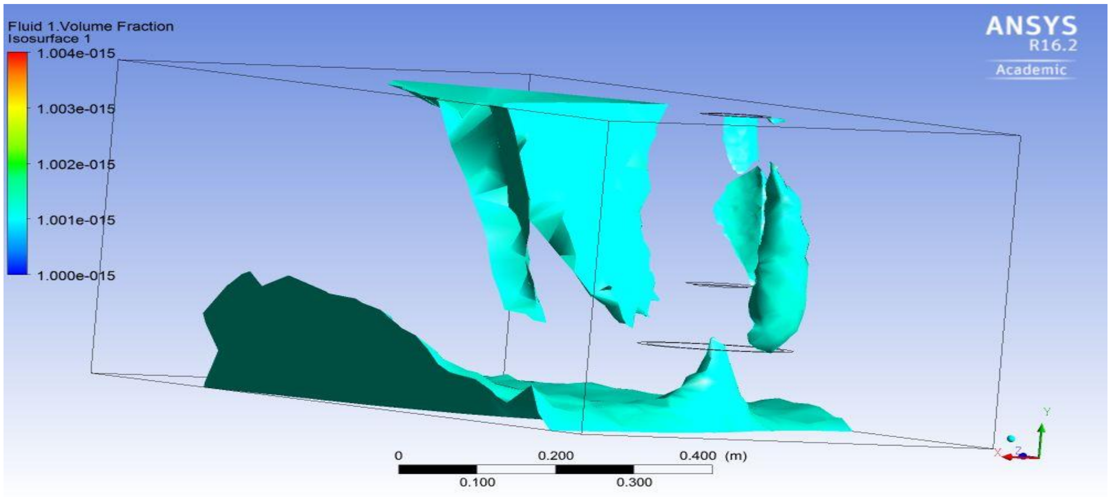


Fig. 11:standard k- Ɛ model

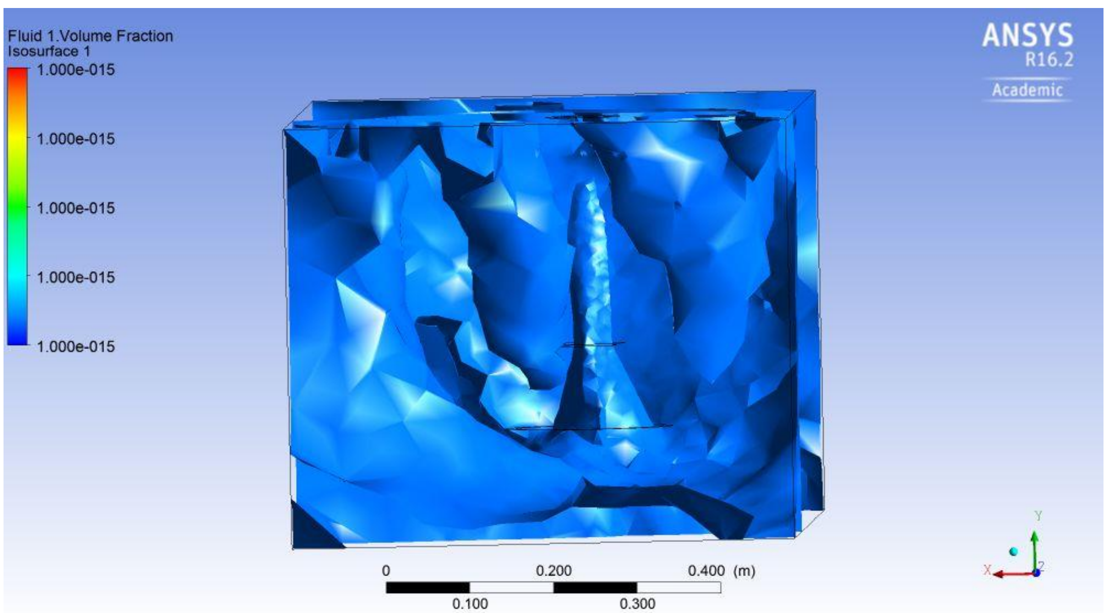


Fig. 12: Reliable K- Ɛ Model

3.Turbulent kinetic energy

Turbulent kinetic energy is great at vortex core region because of the rotation of air-core and the increased flow fluctuation opposes the formation of free surface vortex which explains that strong turbulent fluctuations act like anti-vortex. The following fig. demonstrates the failure in simulation by k- Ɛ model where SST demonstrates a better simulation for same turbulent kinetic energy.

Table 6 :Turbulent Kinetic Energy

MODEL	Turbulent Kinetic Energy(m^2/s^2)	
	minimum	maximum
STANDARD K- Ɛ	3.75 x 10^{-7}	14414.2
RELIABLE K- Ɛ	3.75 x 10^{-7}	1.446 x 10^{4}
SST	3.75 x 10^{-7}	0.226352

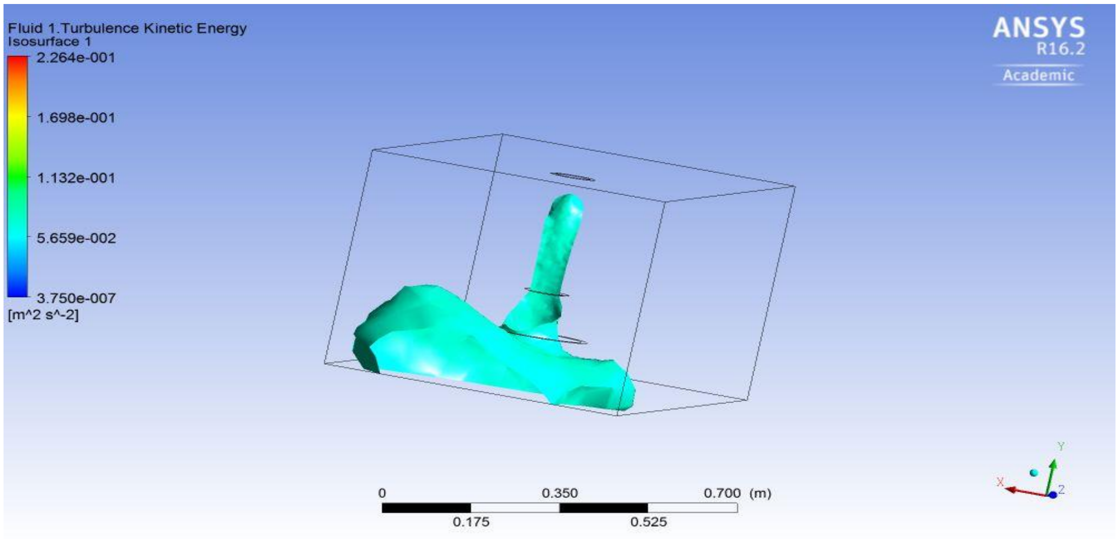


Fig. 13: SST turbulent kinetic energy

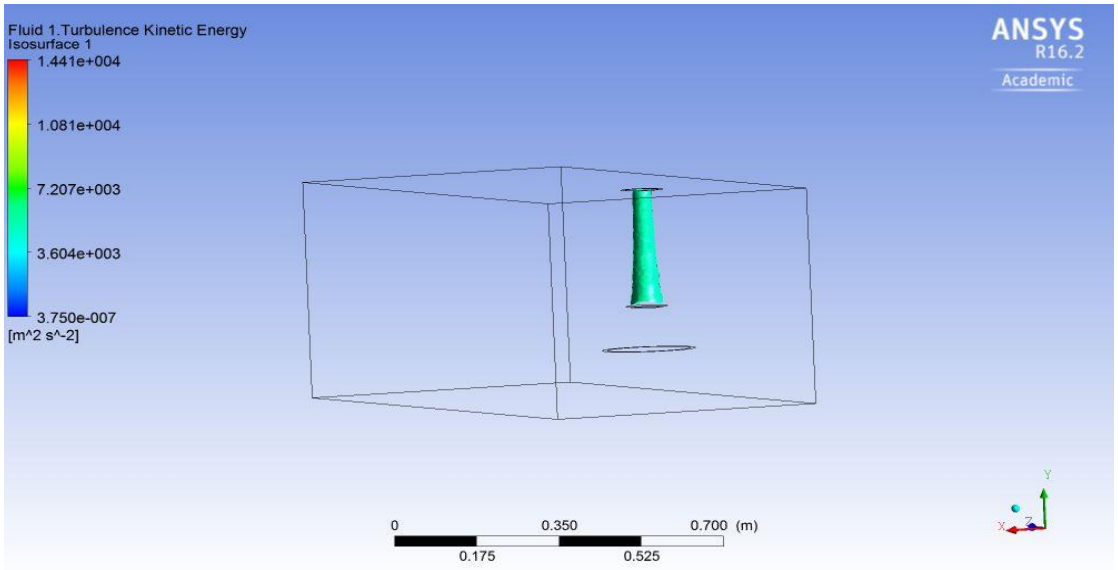


Fig. 14:Standard K- Ɛ Turbulent Kinetic Energy

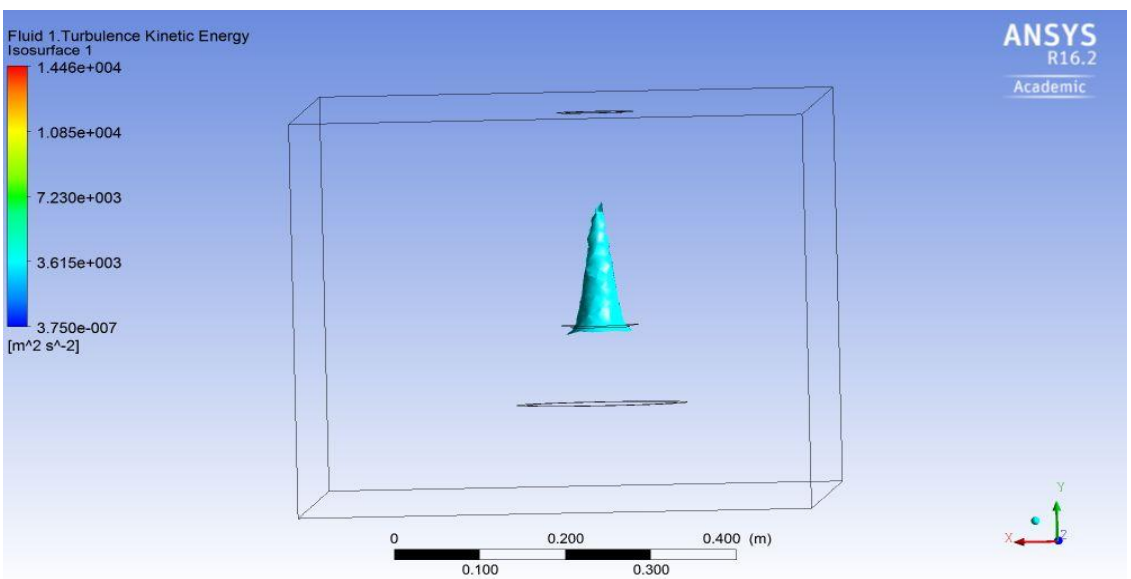


Fig. 15: Reliable K- Ɛ Turbulent Kinetic Energy

4.Turbulent dissipation rate

Turbulent dissipation rate reacts similarly the way turbulent kinetic energy reacts. Turbulent dissipation rate is huge at the core of the air-vortex and also act as anti-vortex because fluctuation due to turbulent dissipation suppresses vertical vortices. In the following fig. k-Ɛ and SST model demonstrates turbulent dissipation rate and it can be seen that simulation is failing in k-Ɛ model.

Table 7 :Turbulent Dissipation Rate

MODEL	**Turbulent Dissipation Rate(m^2/s^3)**	
	minimum	maximum
STANDARD K- Ɛ	8.191×10^{-11}	1.149×10^{2}
SST	8.191×10^{-11}	1.438×10^{9}
RELIABLE K- Ɛ	8.191×10^{-11}	1.445×10^{9}

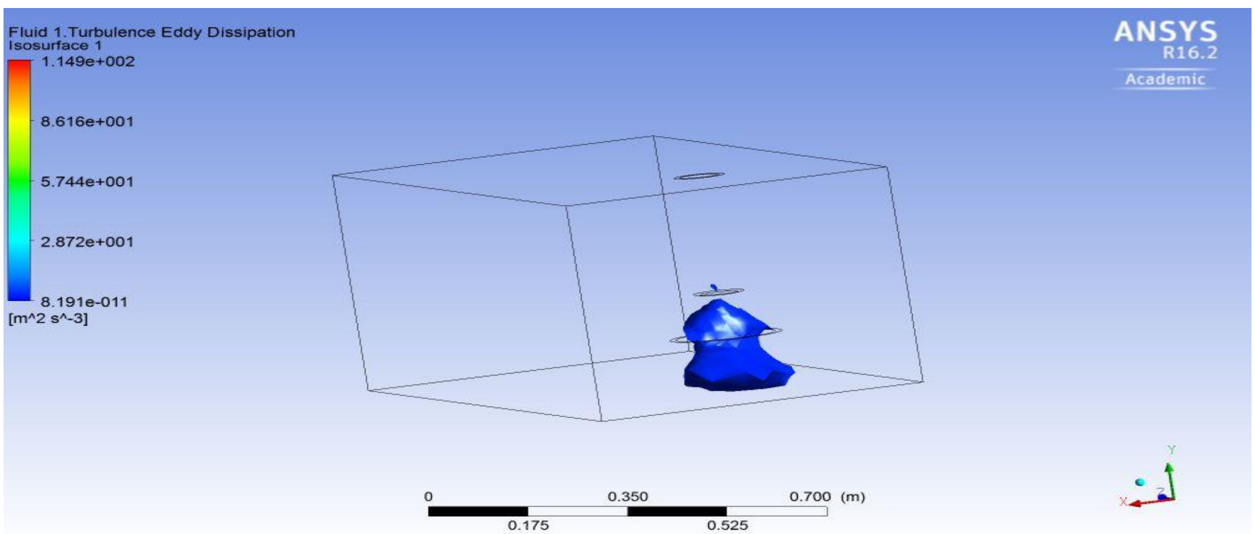


Fig. 16: SST turbulent dissipation rate

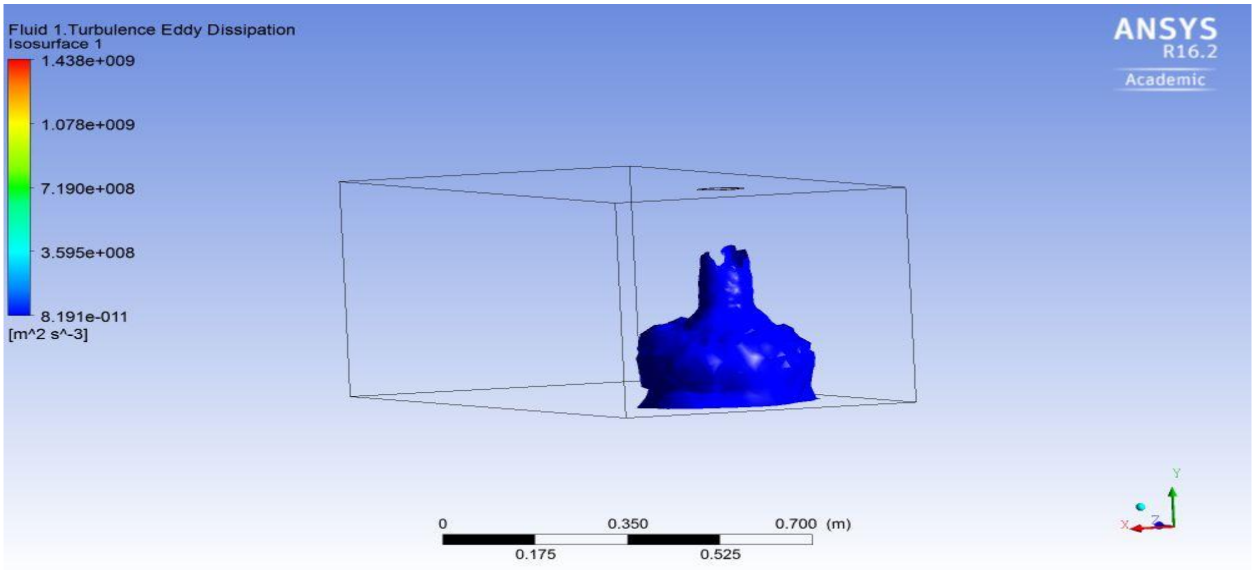


Fig. 17: standard K-Ɛ turbulent dissipation rate

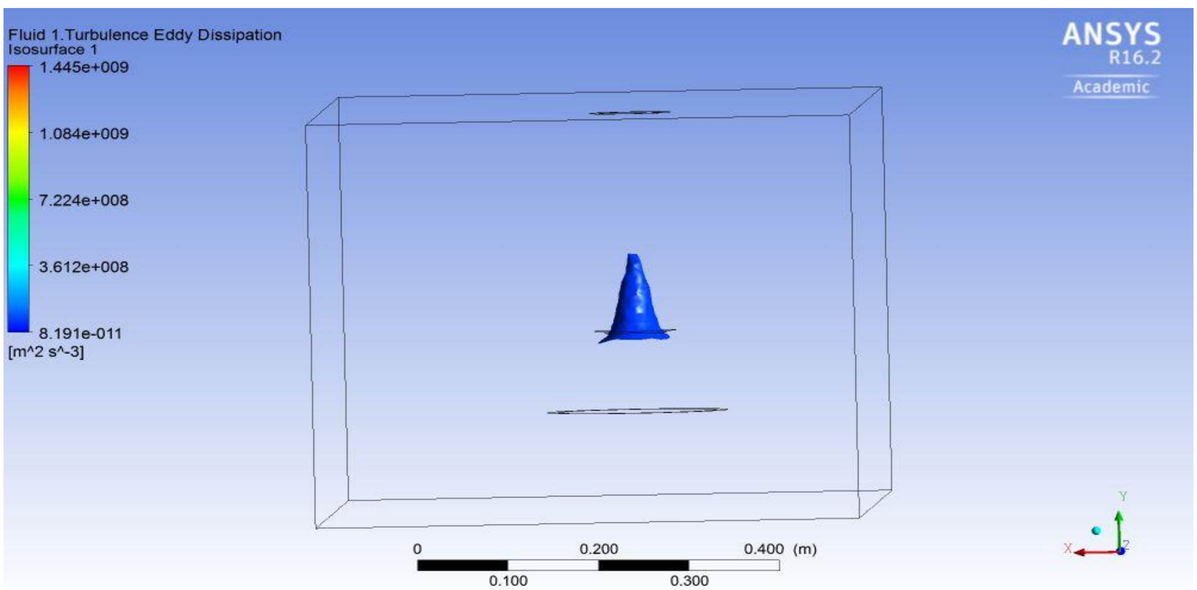


Fig. 18: Reliable K- Ɛ Model Turbulent Dissipation Rate

CONCLUSION

The simulated results shows the comparison between the three model standard k- Ɛ, reliable k- Ɛ and shear stress transport model. It represents that turbulent kinetic energy and turbulent dissipation rate suppresses the formation of vortex. K- Ɛ model shows fail simulation as studied by accumulated time step difference in each model and in turbulent kinetic energy and dissipation rate simulation.

4.1.3 Comparison of CFD Result For Different Submergence Depth:

1. Streamline Flow

Comparison of velocity profile is carried out by applying the above boundary conditions. Under this study we can see the variation of tangential and radial velocity at the suction pipe by varying the submergence depth and comparing the obtained results. In the following figures we can see that the chamber of each suction pipe get affected due to change in submergence depth. There is very little swirl and causes minor disturbance due to difference in submergence depth. The velocity in this case varies from 0 to 49.75m/s. Maximum velocity seems to be at the outflow of the suction pipe and there is uniform variation from inlet to the entrance of pipe intake. Comparison of velocity profile is shown below:

For 300mm Submergence Depth:

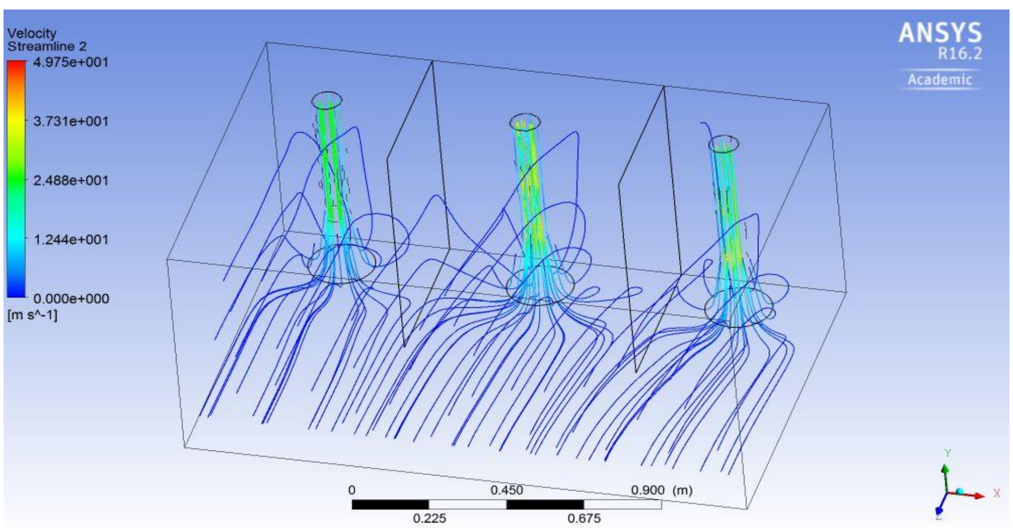


Fig. 19: Streamline Flow D= 300mm

For 130mm Submergence Depth:

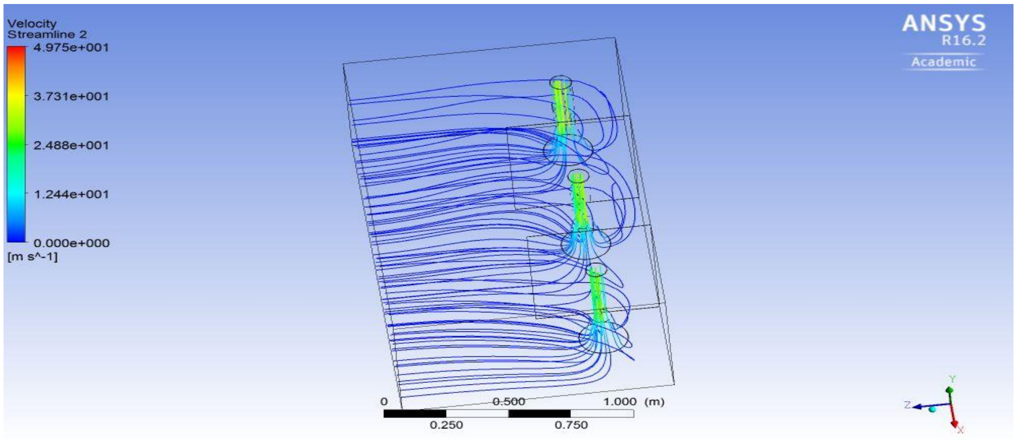


Fig. 20; Velocity Streamline for D=130mm

For 50mm Submergence Depth:

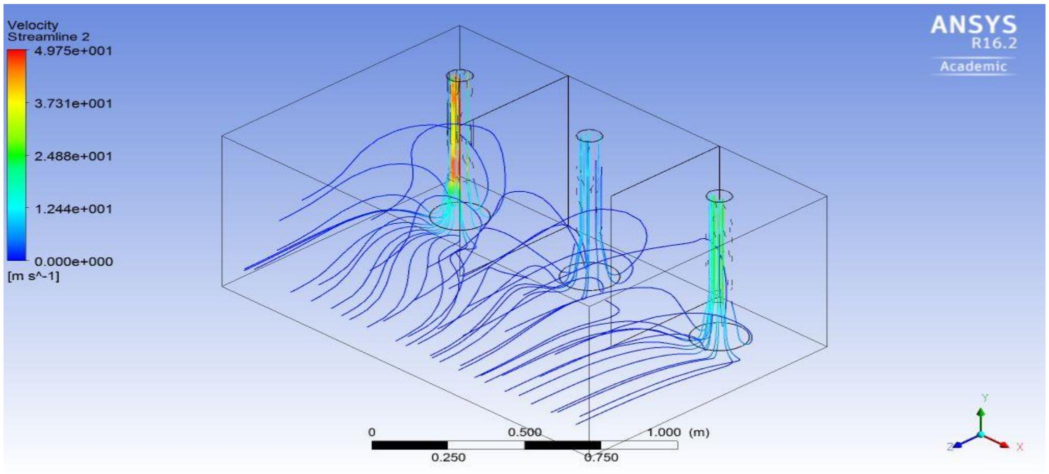


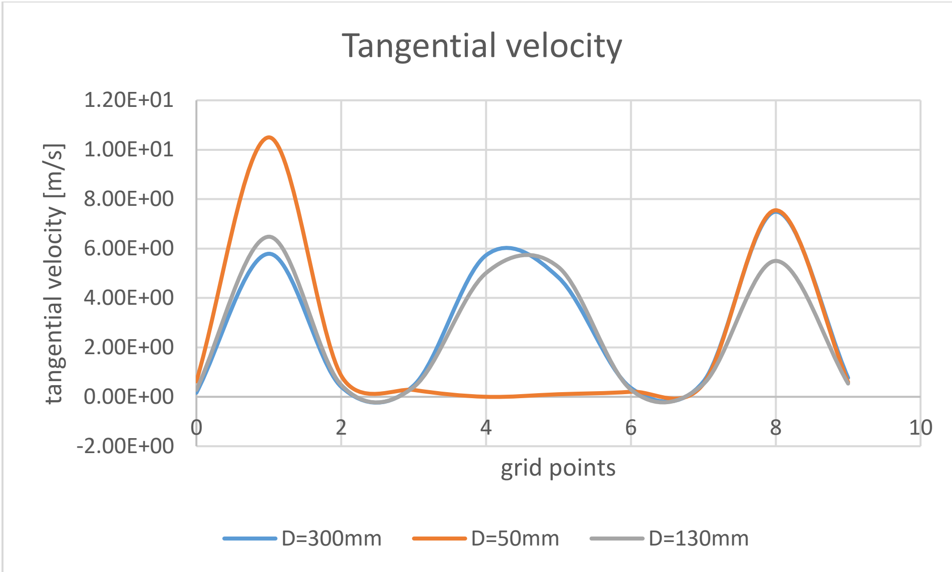
Fig. 21: Velocity Streamline for D=50mm

1.1 Comparison of tangential and radial velocity using simulation model of velocity streamline:
Tangential velocity:

Comparison of tangential velocity is shown below, where the chart demonstrates that there is very minor difference in changing the submergence depth but when the depth is very low than the HI standard which is 50mm there is increase in tangential velocity. From this it can be depicted that with the decrease in submergence depth tangential velocity increases which increases swirling of vortices.

Table 8 : tangential velocity comparison

Grid points	Tangential velocity (m/s)		
	D=300mm	**D=50mm**	**D=130mm**
0	1.64E-01	6.26E-01	2.84E-01
1	5.79E+00	1.05E+01	6.48E+00
2	4.06E-01	8.58E-01	4.80E-01
3	4.56E-01	2.77E-01	3.93E-01
4	5.73E+00	0.00E+00	5.01E+00
5	4.83E+00	1.04E-01	5.25E+00
6	3.62E-01	2.04E-01	2.92E-01
7	6.10E-01	5.14E-01	5.13E-01
8	7.49E+00	7.55E+00	5.50E+00
9	7.67E-01	5.89E-01	5.36E-01



Graphs 1 Tangential Velocity Comparison

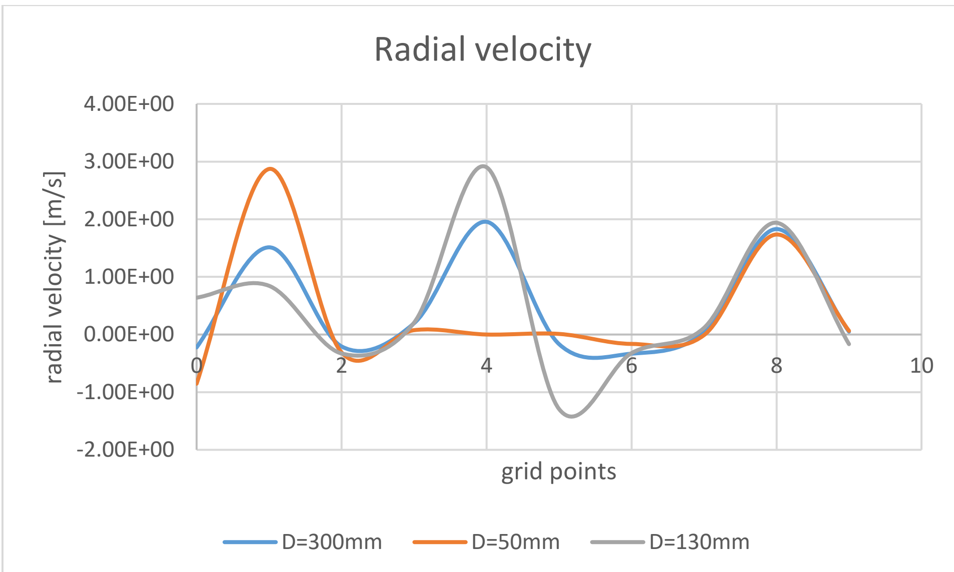
Radial Velocity:

With the increase in submergence depth radial velocity is increasing, which explains that the swirling of vortices will be big but weak in strength that is good for controlling uniformity of approaching flow. Comparison of radial velocity for different submergence depth is shown below:

Table 9 : Radial Velocity

Grid points	**Radial velocity[m/s]**		
	D=300mm	D=50mm	D=130mm
0	-2.24E-01	-8.53E-01	6.39E-01
1	1.51E+00	2.87E+00	8.45E-01
2	-2.07E-01	-3.11E-01	-3.29E-01
3	1.96E-01	7.58E-02	2.06E-01
4	1.96E+00	0.00E+00	2.90E+00
5	-1.67E-01	1.04E-02	-1.29E+00

6	-3.32E-01	-1.63E-01	-3.22E-01
7	4.09E-02	-9.82E-03	1.15E-01
8	1.83E+00	1.73E+00	1.94E+00
9	4.91E-02	6.38E-02	-1.66E-01



Graphs 2 Radial Velocities

2. Turbulent kinetic energy:

Fig. 21 shows turbulent kinetic energy for different submergence depth with the inlet velocity of 0.347 m/s. Turbulent kinetic energy in this case is affecting more the submergence depth of 50mm . Minimum turbulence is 2.74 x 10^{-3} J kg^-1 and maximum turbulence is 3.628 x10^{1} J kg^-1. Minimum turbulence can be seen at the inside of the suction pipeline. Comparison of turbulent kinetic energy is shown below.

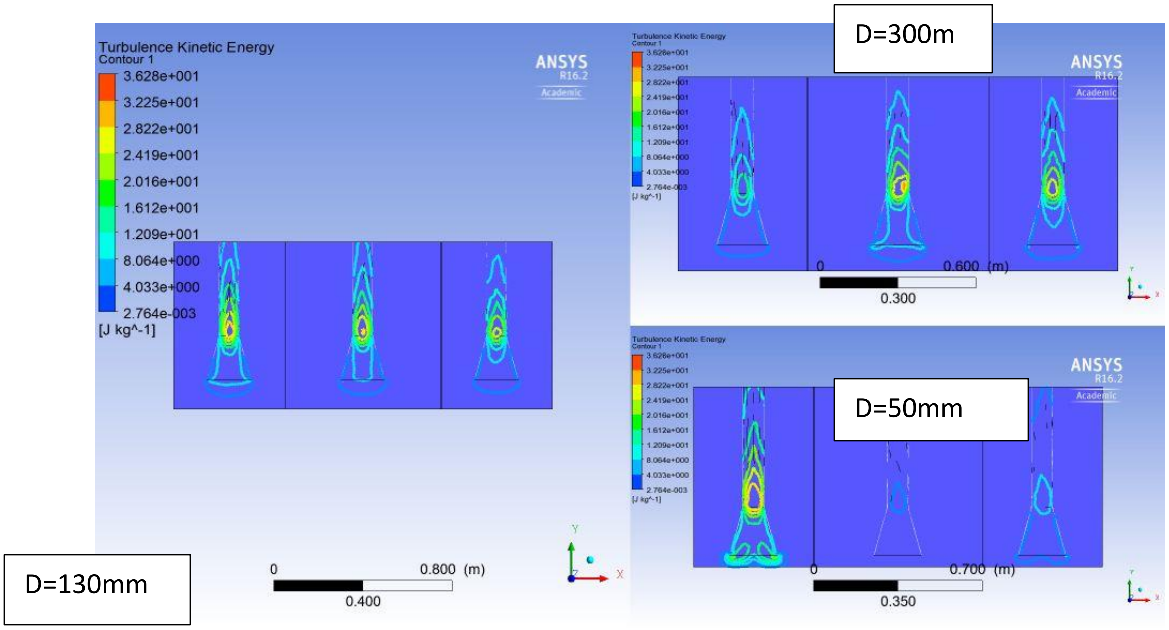
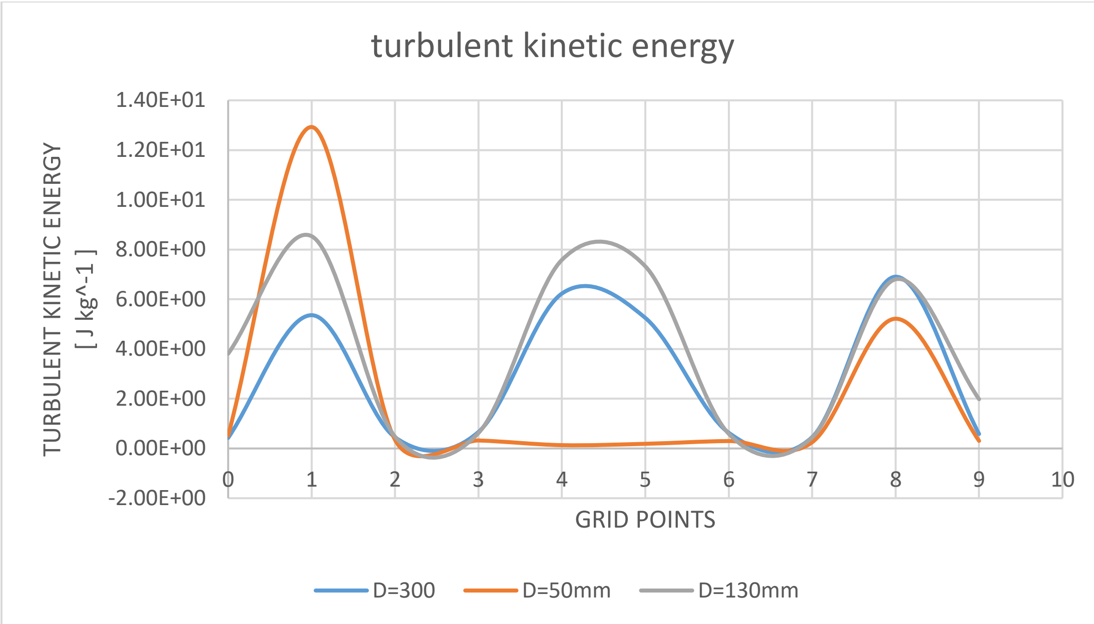


Fig. 22: Turbulent Kinetic Energy

Table 10 Turbulent Kinetic Energy

Grid points	**Turbulent kinetic energy [J kg^-1]**		
	D=300mm	D=50mm	D=130mm
0	4.26E-01	5.24E-01	3.81E+00
1	5.36E+00	1.29E+01	8.54E+00
2	4.51E-01	3.46E-01	4.75E-01
3	6.64E-01	3.22E-01	6.22E-01
4	6.23E+00	1.33E-01	7.58E+00
5	5.25E+00	1.88E-01	7.33E+00
6	6.31E-01	3.00E-01	5.90E-01
7	4.60E-01	2.52E-01	4.44E-01
8	6.91E+00	5.22E+00	6.80E+00
9	5.82E-01	3.06E-01	1.98E+00



Graphs 3 Turbulent Kinetic Energy

3. Turbulent Dissipation Rate:

Maximum turbulent dissipation rate is 6.24 x 10^4 m^2 s^{-3} and minimum turbulent dissipation rate is 2.664 x 10^{-2} m^2 s^{-3}. Variation in turbulent dissipation rate is minor for each difference in submergence depth, but in 50mm submergence depth, it is more than the other two. Turbulent dissipation rate can be seen at the inner side of the wall of the suction pipe. Comparison of turbulent dissipation rate is shown below.

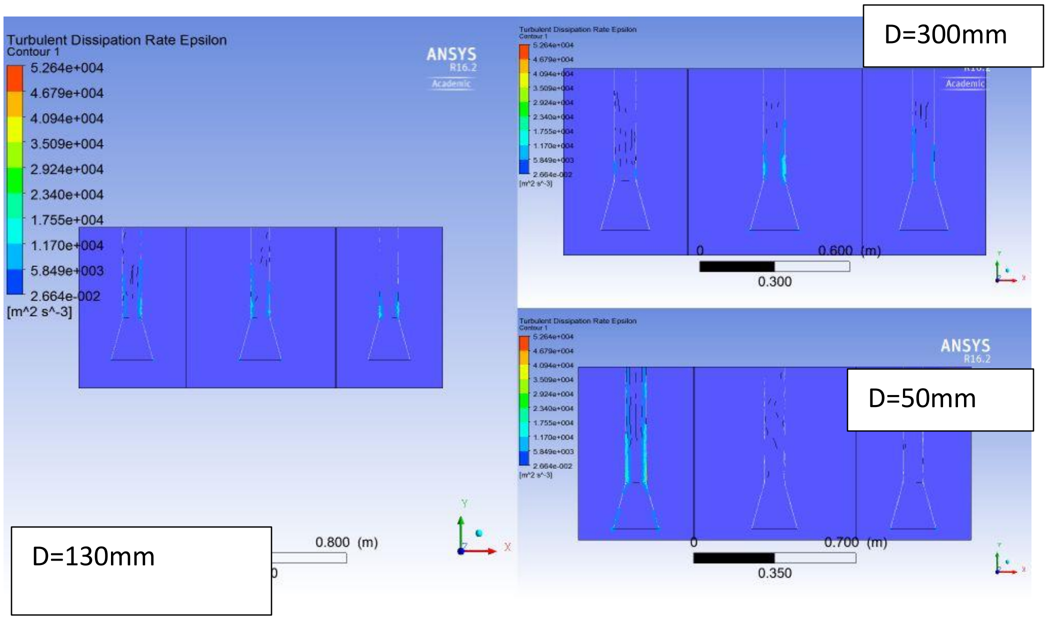
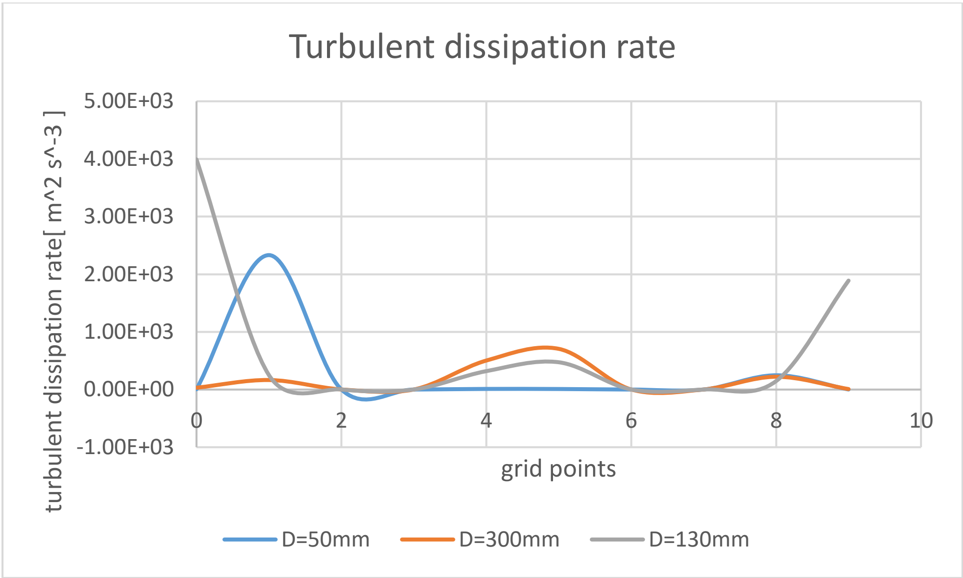


Fig. 23: turbulent dissipation rate

Table 11 Turbulent Dissipation Rate

Grid points	Turbulent dissipation rate ($m^2 s^{-3}$)		
	D=50mm	D=300mm	D=130mm
0	4.65E+00	3.23E+01	3.99E+03
1	2.33E+03	1.65E+02	2.50E+02
2	1.23E+00	1.99E+00	1.71E+00
3	5.33E-01	1.39E+00	1.35E+00
4	1.11E+01	5.03E+02	3.20E+02
5	9.83E+00	7.06E+02	4.74E+02
6	4.79E-01	1.30E+00	1.24E+00
7	7.05E-01	1.62E+00	1.51E+00
8	2.48E+02	2.22E+02	1.47E+02
9	1.29E+00	8.15E+00	1.89E+03



Graphs 4 Turbulent Dissipation Rate

4. Pressure Distribution:

Variation of pressure distribution for different submergence depth is minor . Maximum pressure is 1.016 x 10^5 Pa and minimum pressure is -1.408 x 10^6 Pa. Pressure variation can be seen at intake of the bell mouth pipe where pressure is below zero absolute pressure and the chance of cavitation is much more on those areas. Comparison of pressure distribution can be seen below.

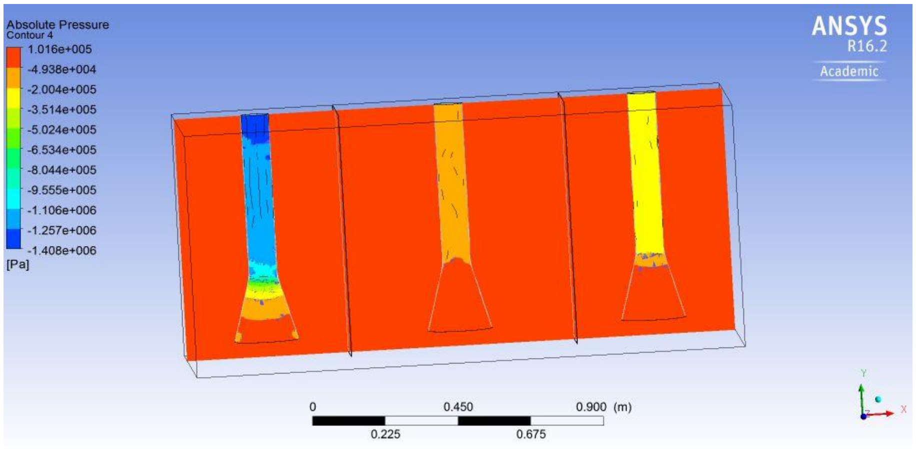


Fig. 24: Pressure Distribution for 50mm

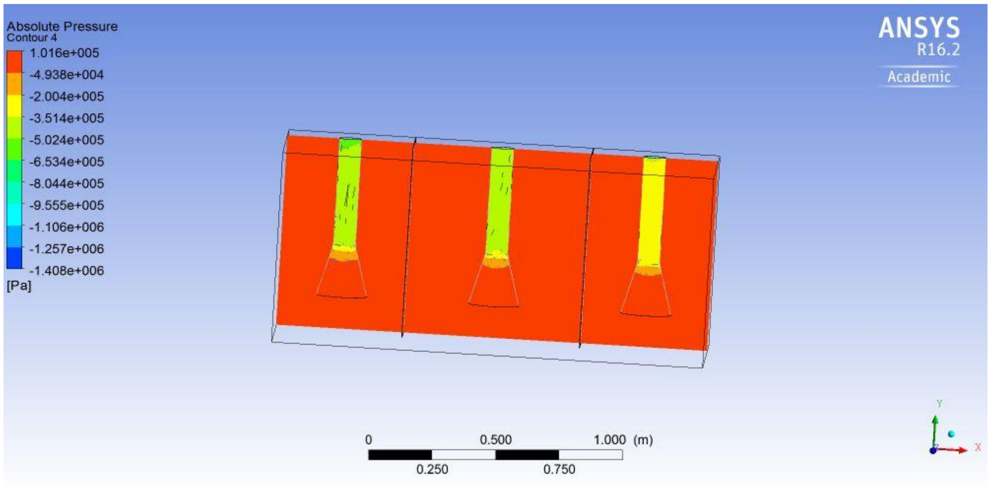


Fig. 25: Pressure Distribution for 130mm

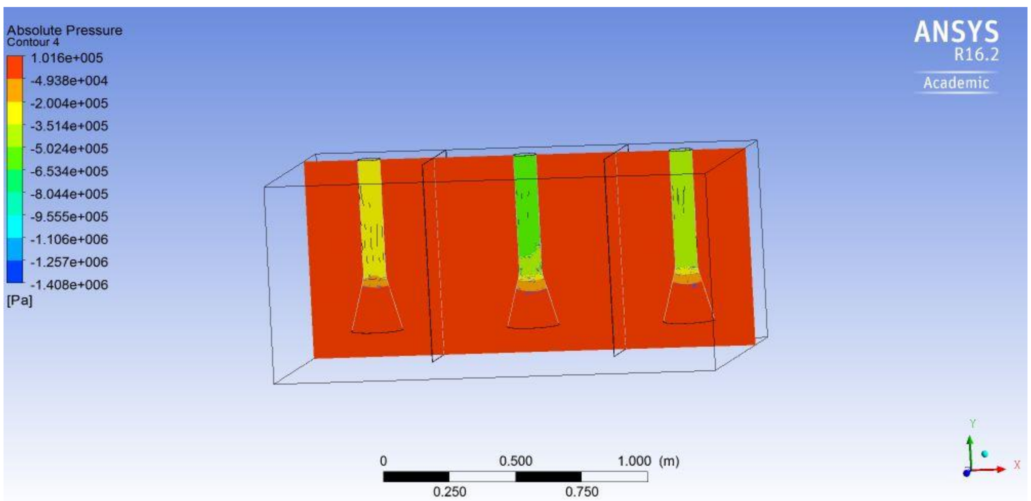
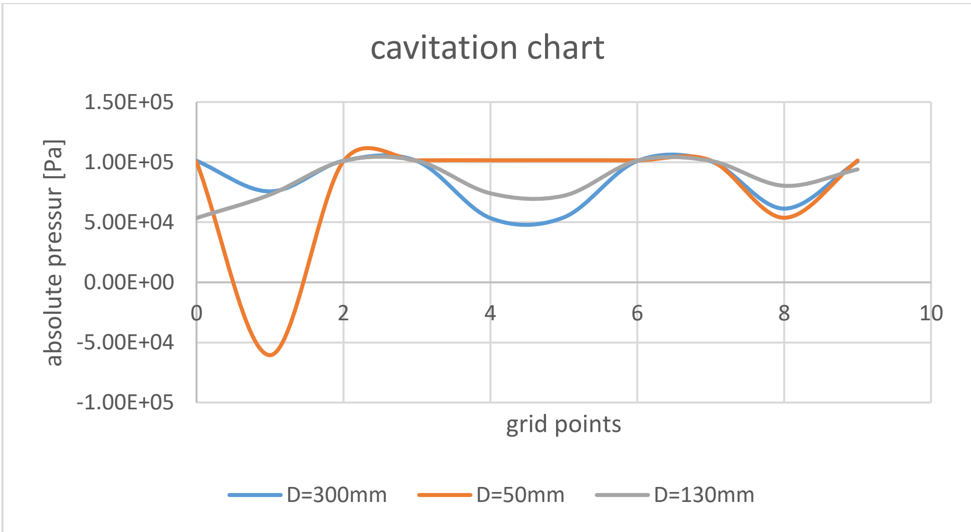


Fig. 26: Pressure Distribution For 300mm

Table 12: Absolute Pressure

Grid points	Absolute pressure [Pa]		
	D=300mm	D=50mm	D=130mm
0	1.01E+05	1.00E+05	5.36E+04
1	7.57E+04	-6.06E+04	7.30E+04
2	1.01E+05	1.01E+05	1.01E+05

3	1.01E+05	1.02E+05	1.01E+05
4	5.35E+04	1.02E+05	7.41E+04
5	5.40E+04	1.02E+05	7.21E+04
6	1.01E+05	1.02E+05	1.01E+05
7	1.01E+05	1.01E+05	1.01E+05
8	6.13E+04	5.37E+04	8.04E+04
9	1.01E+05	1.01E+05	9.40E+04



Graphs 5 Absolute Pressure to Study Cavitation

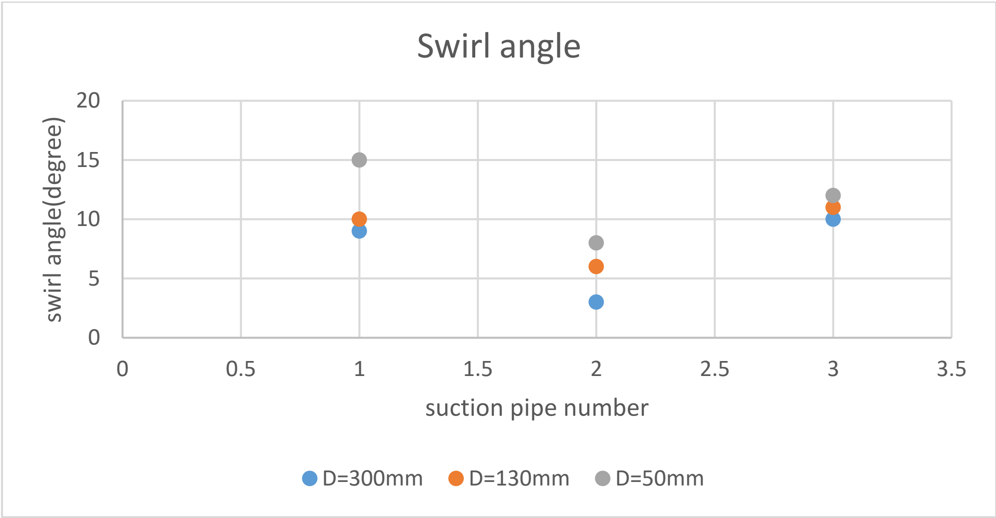
4.1.4 SWIRL ANGLE

Swirl angle calculates the intensity of flow rotation. HI standard is used to calculate swirl angle in experiments that also checks the flow rotation of suction pipe. Rotation time for swirl meter is above 10minutes used for observation. According to HI limit for swirl angle is 5^o. Equation given below is to calculate swirl meter in experiments.

$$\theta = \tan^{-1} \frac{V_\theta}{V_Z}$$

Table 13 : Effect of Submergence On Swirl Angle result obtained from CFD

Submergence depth	Swirl angle ⁰(degree)			
	Pipe1	Pipe2	Pipe3	average
300mm	9^{0}	3^{0}	10^{0}	7.3^{0}
130mm	10^{0}	6^{0}	11^{0}	9^{0}
50mm	15^{0}	8^{0}	12^{0}	11.67^{0}



Graphs 6 Swirl Angle

4.2 CLOSURE

In this chapter use of CFD in the design of suction pipe of a sump pumps is elaborated, especially the software ANSYS FLUENT.The basic working principles of these type of softwares were also discussed.The velocity variation,turbulent kinetic energy and turbulent dissipation rate variation were obtained from numerical simulation for different submergence depth. A comparison of the vswirl angle and location of vortex ,experimentally measured and numerically simulated shows reasonable agreement.

CHAPTER 5

SUMMARY,CONCLUSIONS AND FUTURE WORK

5.1 SUMMARY

In this report, the importance of suction pipe, field applications, functioning, design and their modelling is discussed. Modelling of suction pipe for better pump sump system is necessary Scaled models are constructed and they are analysed for proper intake geometry and modifications are done if required in order to improve the efficiency of prototype. Experimental studies help a lot in order to visualize the flow phenomenon in the prototype and the adverse hydraulic conditions which can occur in the flow. But the construction of the scaled models involve a lot of money and work. Hence now a days numerical studies are gaining a lot of significance in order to make the work easier.

Now a days many software packages are available for the study of fluid flow. All the software packages dealing with dynamics of fluid flow were developed on the basis of Navier Stokes equation. But a lot of assumptions were made in this software by idealising the flow parameters. But considering the difficulty in the construction of the models and expenditure involved these softwares can be used to get nearly correct results. The softwares like ANSYS FLUENT uses meshing methods which will split the whole domain into small volume grids and the facility to provide boundary conditions over the whole domain will nearly simulate real field conditions. In this study FLUENT, has been used to study the model difference, velocity profiles, vortex phenomenon and TKE in steady state conditions.

5.2 CONCLUSIONS

Following conclusions can be drawn from the experimental and numerical studies.

1. While investigating for an effective model, shear stress transport showed better simulation result as standard K-Ɛ model and reliable K-Ɛ model were not able to show better model for volume of fraction. The different characteristics of vortices were considered, where comparison between these three model is done. According to the results obtained, vorticity and swirling strength of the vortices at the vortex core region showed same value but for different time step as for SST model it is t=0.4s , for standard K- Ɛ model it is t=0.2s and for reliable k-

Ɛ model it is t=0.3s,where we can see the failure of simulation for model efficiency. The value of vorticity and swirling strength for which comparisons are done are vorticity for standard k-Ɛ model is 216.345 s^{-1}, and swirling strength 86.55 s^{-1} ; for reliable k-Ɛ model vorticity is 224.981 s^{-1} and swirling strength 89.136 s^{-1} ; and SST model vorticity is 220.981 s^{-1} and swirling strength is 84.874 s^{-1}. It concludes that SST model is better than k-Ɛ models and in between k- Ɛ models , standard k- Ɛ model is better.

2. Further, comparing model for effective simulation result, shear stress transport showed maximum and minimum turbulent kinetic energy and turbulent dissipation rate at the vortex core region, where k- Ɛ models simulation fails and are not able to show complete turbulent kinetic energy and turbulent dissipation rate for the same maximum and minimum value. Vortex core region is a place where we can see vorticity and swirling strength. The turbulent kinetic energy minimum value is 3.75 x 10^{-7} m^2/s^2 which is same for all models and maximum is 0.226 m^2/s^2 for SST, 1.446 x 10^4 m^2/s^2 for reliable k- Ɛ and 14414.2 m^2/s^2 for standard k- Ɛ , if we see in the figures only SST model is showing turbulent kinetic energy around the suction pipe where vortex core region was present whereas k- Ɛ models are showing turbulent kinetic energy inside the suction pipe where there was less vortex core region hence showing fail simulation by them.

3. Comparison of submergence depth for velocity profile showed that tangential and radial velocity for pipe 1, pipe 2 and pipe 3 varies with change in submergence depth. There are very minor differences in change in submergence depth from 300mm to 130mm but there is a fall for 50mm, for a grid point 8 , it can be clearly seen that the tangential velocity for pipe 3 at depth 300mm, 50mm and 130mm is 7.49 m/s, 7.55 m/s and 5.5 m/s; and radial velocity for pipe3 at grid point 8 is 1.83 m/s, 1.73 m/s and 1.94 m/s. with the increase in tangential velocity and decrease in radial velocity the strength of vortex increases for 50mm depth. The minimum submergence suitable according to the velocity profile is 130mm or 300mm because there is very minor difference in between them.

4. The turbulent kinetic energy for pipe 1 at grid point 8 for submergence depth 300mm, 50mm and 130 mm are as 5.36 J kg^-1, 12.9 J kg^-1 and 8.54 J kg^-1. The turbulent kinetic energy

increased in 50mm. the increased turbulence can be seen around the suction pipe.TKE is increasing with the decrease in depth.

5. Turbulent dissipation rate acts similarly as turbulent kinetic energy,dissipation rate for pipe1 at grid point 1 for submergence depth 50mm, 130mm and 300mm are 2330, 250 and 165. It can be concluded that with the decrease in depth turbulence dissipation rate increases.
6. With the increased turbulent kinetic energy and dissipation rate noise, vibration and vortex entrance are some phenomena which will be causing problem for the efficiency of pump, hence minimum submergence depth should be taken which in this report can be 300mm or 130mm as there are very minor differences in the characteristics of vortices for these depths.
7. To consider phenomena of cavitation, pressure distribution in terms of absolute pressure is studied for different submergence depth, hence for pipe 1 at grid point 1, absolute pressure for submergence depth 300mm is 7.57×10^4, 130mm is 7.30×10^4 and for 50mm is -6.06×10^4. 50mm is the only depth where absolute pressure is below zero, this is the condition for cavitation causing boiling of water, which should be avoided hence 50mm depth or in general depth very near to the ground clearance is not suitable for suction pipe functioning and for better efficiency of pump.
8. The comparison of swirl angle for different submergence depth is done. For which we can see that swirl angle decreased with increase in depth as this depth should be the minimum submergence depth. For submergence depth 300mm, 130mm and 50mm swirl angle is 9^{o}, 10^{o} and 15^{o}. Swirl angle should be less than 5^{o} (HI standard guide) for the less affect of vortex flow, though in this report there is none found less than 5^{o}.
9. Location of vortices and swirl angle were compared. Flow in CFD showed free surface vortices, side wall vortices and floor vortices. The vortices do vary for different submergence depth as the swirl angle decreased with the increase in submergence depth. To validate the model comparison of model with data obtained from the experiments of Korea university[1] is done. Which can be concluded that CFD model is adequate and compatible to work with as the swirl angle obtained for pipe1 through experiments is 10.7^{o} and for CFD it is 10^{o}..
10. At normal water level, the performances of the suction pipe and sump chambers are satisfactory to provide sufficient water supply to the pumps for 130mm and 300mm submergence depth whereas for 50 mm submergence depth it was unsatisfactory because of non-uniformity in flow causing noise, vibration and cavitation.

11. Numerically simulated results are obtained by considering many idealistic flow parameters and hence the results may not match exactly with that of experimental ones.We can see error in the charts which can be concluded that there were fluctuation in the designing of the model.

5.3 MEASURES TO PREVENT VORTEX FLOW

- Baffle plate: These are designed to direct the flow and act as obstructing vanes. It also helps in preventing from the effect of vibration.
- Splitter plate: Vortex shedding is controlled by using splitter plate which diverts boundary layer away from the intake.
- Partitioned structures : Open sumps condition gets better when some piers like dividing walls are placed between the multiple suction pipe of pumps in a single intake structure. We can see turbulence in flow if it is not divided by walls. Piers are the dividing wall which is necessary for pumps with discharge of 315 l/s.
- Trash racks and screens: Partially clogged trash racks or screens can create severely skewed flow patterns. If the application is such that screens or trash racks are susceptible to clogging, they must be inspected and cleaned as frequently as necessary to prevent adverse effects on flow patterns.
- Minimum submergence depth: Depth of the suction pipe in sump should be at such height that it creates less noise and vibration and air entrainment should be less for better efficiency of the pump.

5.4 SCOPE OF FUTURE WORK

Following are the the proposed future work

1. To carry out more experiments on bell throat pipe intake system, to conduct sediment transport studies in the models.
2. To carry out sensitivity analysis for the meshing using FLUENT to improve the accuracy of the numerical results.
3. To perform transient state simulations in the FLUENT and study the flow phenomenon.

REFERENCES

1. C. G. Kim, B. H. Kim, B. H. Bang and Y. H. Lee (2014)," Experimental and CFD analysis for prediction of vortex and swirl angle in the pump sump station model", Journal of Institute Of Physics,2014

2. ANSYS, 2013, Fluent theory guide, Version 13

3. Arboleda. G and Fadel (1996)," Effect of approach flow conditions on pump sumps", Journal of hydraulic engineering,1996,vol 122,489-494

4. Constantinescu G.S., and Patel, V.C. (1998), "Numerical model for simulation of pump-intake flow and vortices", ASCE Journal of Hydraulic Engineering, Vol.124, No.2, 123-124.

5. Constantinescu1, G.S. and Patel, V.C. (2000), "Role of Turbulence Model In Prediction of Pump-Bay Vortices", ASCE Journal Of Hydraulic Engineering, Vol.126, No.5, 387-390.

6. Choi Joi Wong, Young Do choi, Chang goo kim (2010),"Flow uniformity in a multi intake sump model" ,Journal of mechanical science and technology,2010,1389-1400.

7. Desmukh & Gahlot (2010)" Simulation of Flow through a Pump Sump & its Validation" International Journal of Recent Research and Applied Studies (IJRRAS) in July 2010, Volume 4 issue 1,164-176.

8. Fluent Manual,2012

9. Gerard bois and Abir Issa (2010) "Numerical simulation of flow field in water pump sump and inlet suction pipe " 25th IAHR symposium on Hydraulic machinery and Systems, September 2014,Timisoara,Romania,272-281.

10. Hai feng and hong xun (2008)," Experimental and Numerical Investigation of Free Surface Vortex" Journal of Hydrodynamics. 20(4) 485- 91

11. Hydraulic Institute Standard, 1998, American National Standard for Pump Intakes Design, ANSI/HI 9.8.

12. Hong xun and Jia hong (2007),"Numerical simulation of 3D turbulent flow in the multi intake sump of the pump station", Journal of hydrodynamics, vol 19,42-47.

13. Issa A, Bayeul-Lainé A C, Bois G (2008) Numerical Simulation of Flow Field Formed in Water Pump-Sump 24th Symp. Hydraulic. Machinary. Systems (Foz de Iguacu, Brasil

14. Jie min Zhan and beng chang wang (2012),"Numerical investigation of flow patterns in different pump intake systems", Journal of hydrodynamics, vol 24, 873-882.

15. .Johanason.E and White.E (2005)"Advancement in hydraulic modelling of cooling water intakes in power plants " Proceedings of PWR2005 ASME Power Chicago, Illinois

16. .Kim.C.G. Kim B.H, Y.H.Lee, (2014) "Experimental and CFD analysis for prediction of vortices and swirl angle in pump station model", International symposium of cavitation and multiphase flow,ISCM 2014.

17. Meena vikas, Ajay Bhat and Eldho T.l (2013)"Numerical simulation of pump sump flow characteristics " Journal of Engineering and Technology. Volume 2 , Issue 3 ,July – September, 2013

18. Nagahara, Toshiyuki Sato, Tomoyoshi Okamura,(2003)"Measurement of flow around the submerged vortex cavitation in a pump intake by means of PIV", Fifth international symposium on cavitation, Osaka, Nov 1-4,2003.

19. Nakato. T 1989 A Hydraulic-Model Study of the Proposed Pump-Intake and Discharge Flume Crystal River Cooling-Tower Project (IIHR Rep. No.339, Iowa Inst.of Hydr., the Univ of Iowa. Iowa City)

20. Padmanabhan. M and Hecker. E (1984) , "Scale effects in pump sump models", Journal of hydraulic engineering,1984,vol 110,1540-1556

21. Pratap K.M. and Chavan. D.S (2013)"CFD Analysis of flow in pump sump to check suitability for better performance of pump " International Journal of Mechanical Engineering and Robotics, Volume 1,issue 2,283-300

22. Shazy.A and Shabayek, (2010)"Improving approach flow hydraulics at pump intakes ",International journal of civil and environmental engineering"2010,Vol 10,No 6.

23. Shin, C.S., E.M. Greitzer, W.K. Cheng, C.S. Tan, and C.L. Shippee, (1986), "Circulation measurements and vortical structure in an inlet-vortex field", Journal of Fluid Mechanics, Vol 162, Pp 463-487

24. Tomoyoshi and Jun Matsui (2007)"CFD prediction and model experiment on suction vortices in pump sump" The 9th Asian conference on fluid machinary October 16 -19, 2007,,Jeju, Korea.

25. **Text books**

- Turbulent flows by G.viswas and E.Eswaran (2002) published by Narosa.
- Open channel flow by Subramanyam. K (2009),Tata Mc.Graw hill
- Fluid mechanics by R.K.Bansal (2005)
- Vertical turbine,mixed flow and propeller pumps by J.L.Dicmas, published by Tata MC graw hill