Evaluating Performance of Centrifugal Pump through CFD while Modifying the Suction Side for Easting Discharge

Miss. Vibha P.Pode, Prof.S.V.Channapattana, Prof.A.A.Bagade

Mechanical engineering,Student
Mechanical engineering, Rajashri Shahu College of Engineering, Tathavade
Mechanical engineering Department, D.Y.P.S.O.E.A Ambi

Abstract- The conventional suction geometry is not efficient for higher capacity of pump and thus reduced discharge on the delivery side. Intake manifold is being designed for this work. The previous configuration would be studied using CFD techniques while pursuing the objective of arriving at the most efficient geometry for the given application. The marginal increment in the discharge for the Centrifugal Pump tends to depreciate with each marginal rise in capacity the pump; especially for the higher order pumps (25HP and above).

The prominence of vortices along with turbulent flow at the regions in the suction pipe affects the flow of water and consequently the discharge. The discharge the `sump' is not favorably designed for aiding the intake through the suction pipe. This work would focus on Design alternatives for minimizing the vortices within the suction pipe and enhancing the discharge through possible use manifold at the suction end. Alternatively, efforts would be pursued for addressing the Design of the Sump (Tank) for facilitating the flow of water at the suction end while smoothing out the in pipe.

Index Terms- alternative geometry, pressure drop, suction side, performance of pump, flow analysis

I. Introduction

Suction or inlet condition is an important factor of pump hydraulic system design. Due to cavitations, vortices, and prime losses, no proper allowances may occur. Pressure which is created by pump is lowered at the suction nozzle which induces the fluid to enter through inlet piping. Less liquid being handled if any design that slow down the efficient transport of this liquid. In some cases Physical damage or crackers to the pump or any other part of pump is because of poor performance and bad designs of pump.

The study of pump suction system configuration is classified into two parts: (1) suction piping and (2) suction source. For properly design an efficient system, critical consideration should be given to suction piping and suction source. This study deals with pump suction system configuration and is applicable to all types of pumps, and successful operation and useful life of every pump can be so dependent on a properly designed and thought-out suction arrangement.

The main aim of sump is to provide water with uniform velocity during the pump operation, abnormal flow phenomena such as cavitations, flow separation, pressure loss, vibration and noise occur often by flow unsteadiness and instability. Especially, free and subsurface vortices containing air occurred in sump pumps seriously damage to pump system. According to the HI standard of Hydraulic Institute or JSME criteria for a pump sump design, these vortices should be prevented and their disappearance must be verified by model test in the construction of pump station.

To reduce these vortices and for the advanced pump sump design with high performance, it is very important to know the detailed flow information in sump system. For this purpose, to date many researchers have made experimental and numerical studies on the flow in pump-sump Now days it is common practice in India, that while engineering big lift Irrigation , Drinking water scheme or Boiler feed pump unit ,Sump model study is conducted to confirm suitability of pump sump , also parallel CFD analysis is done to check suitability of sump. Initially while engineering pumping scheme general guidelines of appropriate standards HIS or BHRA are used to decide the sump dimensions and then sump model study is conducted in presence of manufacturer and customer representatives. The flow conditions at entry to a pump depend upon flow conditions in approach channel, sump geometry, location of pump intake with respect to the walls, velocity changes and obstructions such as piers, screens etc., and rotational tendencies in flow produced upstream of the pump bays. Analytical determination of the flow conditions in a sump is not an easy task due to the complex nature of the flow. Moreover the analytical solution may not completely predict the actual conditions in the sump due to the assumptions made for simplifying the analysis. Thus at present model studies are the only tool for developing a satisfactory design of a pump sump, yet numerical simulation is a very good facility for reducing the time and cost involved in the design process.

II. Literature Review

S. R. Shah, et al. [3] conducted various experiment in laboratory to validate a numerical model developed to simulate the three-dimensional turbulent flow in a water pump intake bay. The experiments were conducted on the basis of flow visualizations and measurements with particle image velocimetry.

A numerical model was developed by Constantinescu and Patel (1998 a,b) to simulate the flow in water intake bays. This paper describes comparisons between the experimental and numerical results as there was no experimental data that was complete and comprehensive enough to validate the numerical model. The location, number, and general structure of the subsurface and free-surface vortices prophesied by the numerical model were approximately correct with those observed in the experiment. The only blemish in the numerical model was that the exception of the strongest vortex, attached to the nearest sidewall, which was prophesied, was more dispersed and less intense than the vortices observed in the experiment. The result propound that the numerical model which could be executed to preliminary design to identify geometric configurations and flow parameters that may lead to strong vortices in the intake and swirl in the suction column.

P.S. Mahar, R.P. Singh [4] reported that the total cost of a pumped water supply system includes the capital costs, the replacement costs, and the energy costs concomitant to the system's operation and pumping units. The optimization model assists to minimize the total annual cost of the pumping main and pump, satisfying the pump characteristic curve equations. This model determines the optimal diameter of pumping main with pump efficiency for a required discharge for an available diameter.

III. METHODOLOGY

[A] Mathematical Model: In mathematical model, the empirical formulae in the Engineering domain can be applied to get the desired solution for the problem. There are two methods to calculate the pressure drop, namely; the Bisection method and the Modified Newton Raphson method. In our case we will use the Modified Newton Raphson Method.

[B] Analytical Model: This is nothing but the computational method. This method provides the simplicity and helps us to solve the problem with robustness. The software's used will be in FEA (Finite Element Analysis) domain. The analytical model is then further divided in three parts namely; Preprocessing, Solving, Post processing. The preprocessing can be done in modelling software such as Catia and then followed by solver GAMBIT/ANSYS and post processing will be in ANSYS FLUENT. Here the result of the mathematical model will be compared with the result obtained from the computer.

IV. CFD STUDIES

Definition of CFD

"The science of predicting fluid flow, heat transfer, mass transfer (as in perspiration or dissolution), phase change (as in freezing or boiling), chemical reaction (e.g. combustion), mechanical movement (e.g. fan rotation), stress or deformation of related solid structures (such as mast bending in the wind), and the related phenomena by solving mathematical equations that govern these processes using a numerical algorithm on a computer" (FLUENT, 2006)

Computational fluid dynamics (CFD) uses numerical methods to solve the fundamental nonlinear differential equations that describe fluid flow (the Navier- strokes and allied equations), for predefined geometries and boundary conditions. The result is a wealth of the predictions for flow velocity, temperature, density, and chemical concentrations for any region where flow occurs. CFD analysis begins with a mathematical model of a physical problem, conservation of matter, momentum, and energy must be satisfied throughout the region of interest. Fluid properties and modeled empirically. Simplifying assumptions are made in order to make the problem tractable (e.g., steady-state, incompressible, in viscid, two dimensional). Provide appropriate initial and boundary conditions for the problem. CFD applies numerical methods (called discretization) to develop approximations of the governing equations of fluid mechanical in the fluid region of interest. The solution is post processed to extract quantities of interest (e.g. lift, drag, torque, heat transfer, separation, pressure loss, etc.).

Basics of Computational Fluid Dynamics:

Computational Fluid Dynamics (CFD) is a computer-based tool for simulating the behavior of systems involving fluid flow, heat transfer and other related physical processes. It works by solving the equations of fluid flow over a region of interest, with specified (known) boundary conditions for that region. CFD is a sophisticated computationally-based design and analysis technique. CFD software gives you the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modeling. This software can also build a virtual prototype of the system or device before can be apply to real world physics and chemistry to the model and this software will provide with images and data which predict the performance of that design. Computational fluid dynamics (CFD) is useful in a wide variety of applications and used in industry. CFD is one of the branches of fluid mechanics that uses numerical methods and algorithm can be used to solve and analyze problems that involve fluid flows and also simulate the flow over a piping, vehicle or machinery. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. More accurate codes that can accurately and quickly simulate even complex scenarios such as supersonic and turbulent flows are ongoing research. Onwards the aerospace industry has integrated CFD techniques into the design, R & D and manufacture of aircraft and jet engines. More recently the methods have been applied to the design of internal combustion engine, combustion chambers of gas turbine and furnaces also fluid flows and heat transfer in heat exchanger. Furthermore, motor vehicle manufactures now routinely predict drag forces, under bonnet air flows and surrounding car environment with CFD. Increasingly CFD is becoming a vital component in the design of industrial products and processes. This study deals with the study of water outlet temperature from helical coil system. Computational fluid analysis is carried out to determine the temperature profile. Computational fluid analysis is carried using software CATIA V5 which is used to build the model and mesh it in ICEM CFD and ANSYS FLUENT is used to carry out the temperature analysis. This total analysis is known as Computational fluid dynamics analysis. Today, well tested commercial CFD packages not only have made CFD analysis a routine design tool in industry but also helping the research engineer in focusing on the physical system more effectively. Before doing the analysis it is important to have an overview of what fluent is and how does it work.

Experiments vs. Simulations

Experiments	Simulations
Quantitative description of flow phenomena using measurements • for one quantity at a time • at a limited number of points and time instants • for a laboratory-scale model • for a limited range of problems and operating conditions Error sources: measurement errors, flow disturbances by the probes	Quantitative prediction of flow phenomena using CFD software • for all desired quantities • with high resolution in space and time • for the actual flow domain • for virtually any problem and realistic operating conditions Error sources: modeling, discretization, iteration, implementation

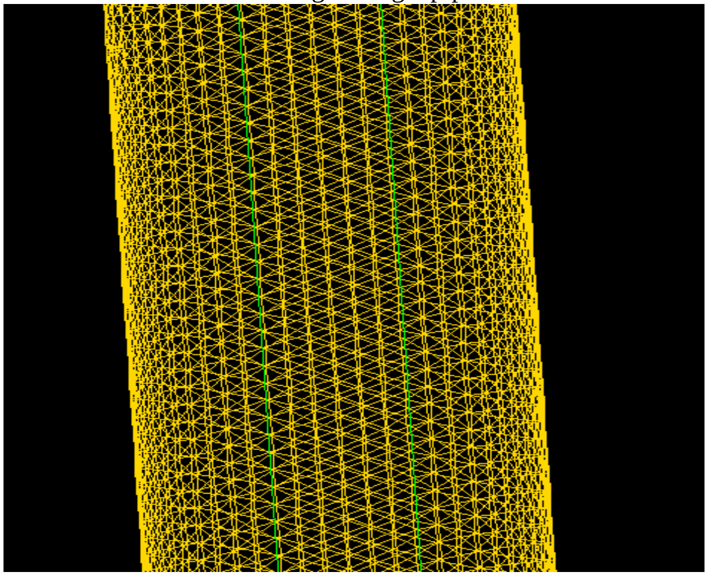
V. RESULT OF CFD ANALYSIS

A number of computations by the CFD analysis were carried out for various conditions using a desktop personal computer. One case of computation took eight hours to get animation for flow visualization. In the following analysis, water is considered as working fluid. The CFD software provided animation video files for visual understanding of flow pattern.

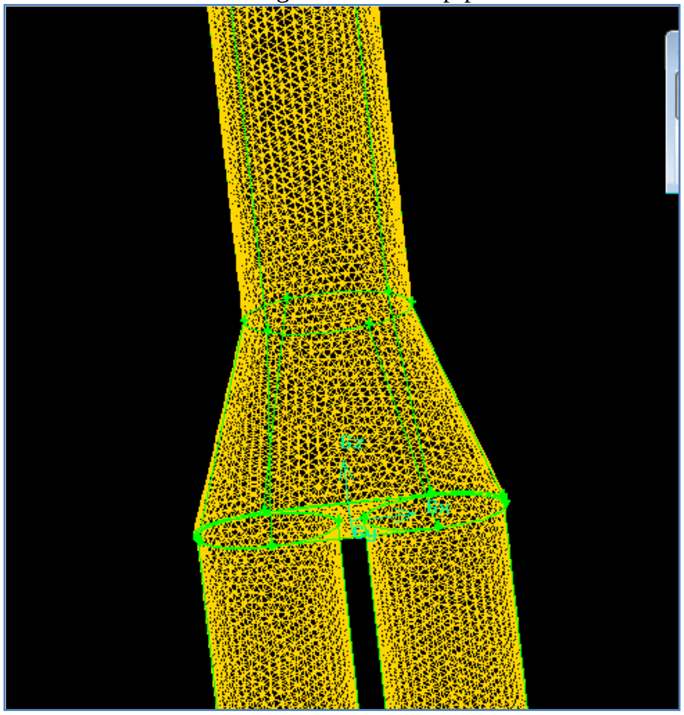
The results of the computational simulation can be analyzed using number of variables. In this study it has been restricted to the comparison of results based on the pattern of streamlines of flow and the velocity profiles. The streamline pattern shows that a very large rotating mass of fluid is created in the forebay portion but it not affects the smooth entry of water in each pump due to the length of the forebay is more. Also there flow is maximum on the side where space for future expansion is kept blank, similarly in experimental we get the maximum flow on blank pump chamber side which is analyzed by using wool thread which is shown in figure 5. And there is no any surface or submerged vortices in the pump chamber.

Meshing detail
Meshing type : tetrahedral
Element size : 5

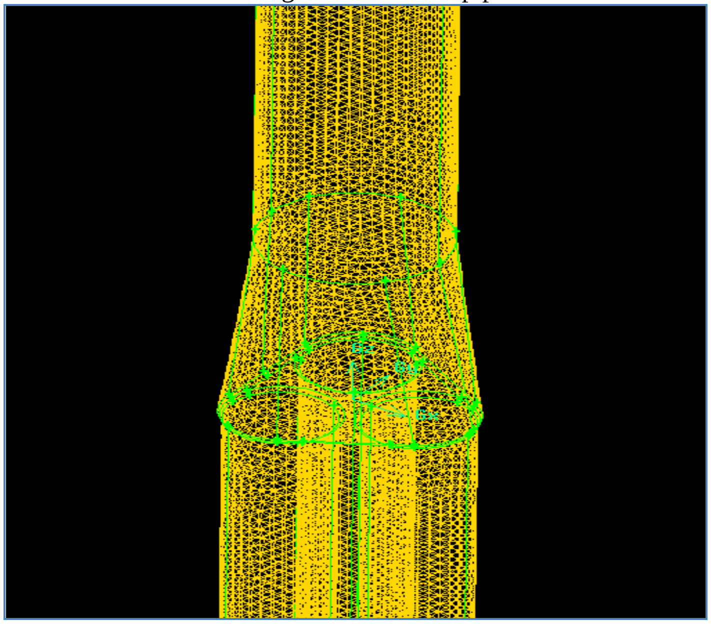
Meshing of single pipe



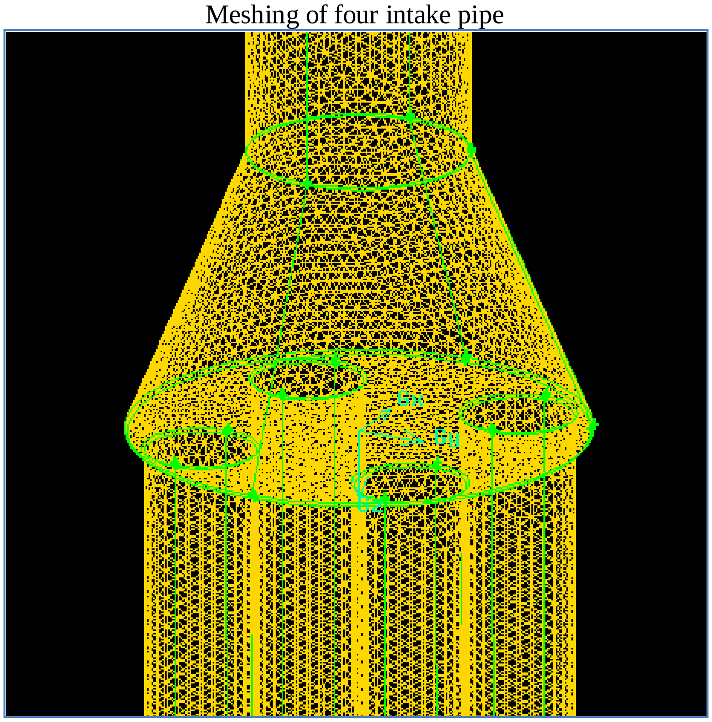
Meshing of two intake pipe



Meshing of three intake pipe



Meshing of four intake pipe



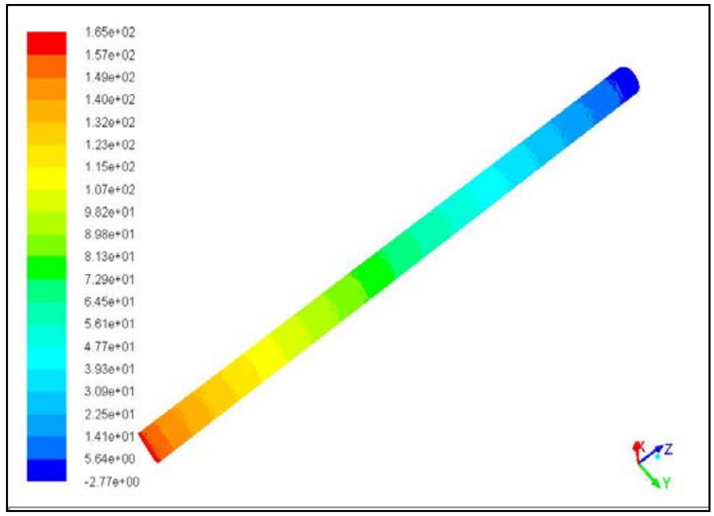
A. Pressure Distribution

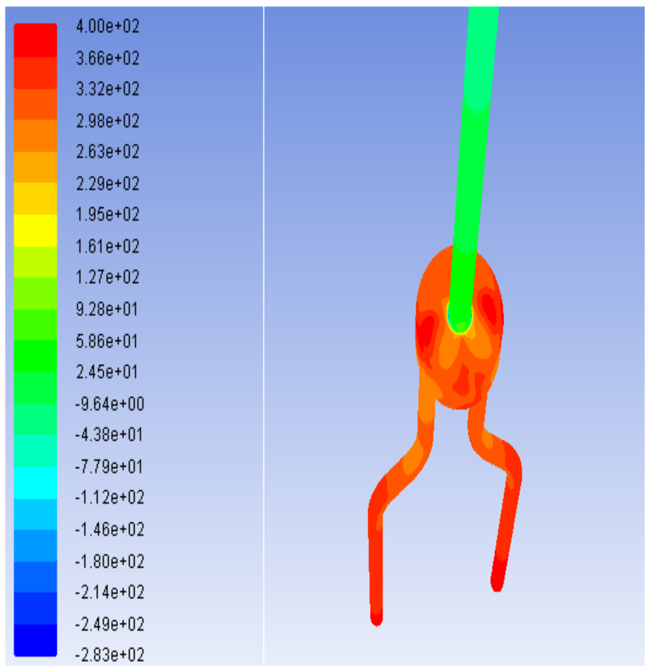
Pressure distribution calculated by the CFD is shown in figures. Red colour indicates pressure level is maximum and blue is indicates pressure level is minimum. In the figure, For single pipe pressure is maximum at inlet and minimum at outlet.

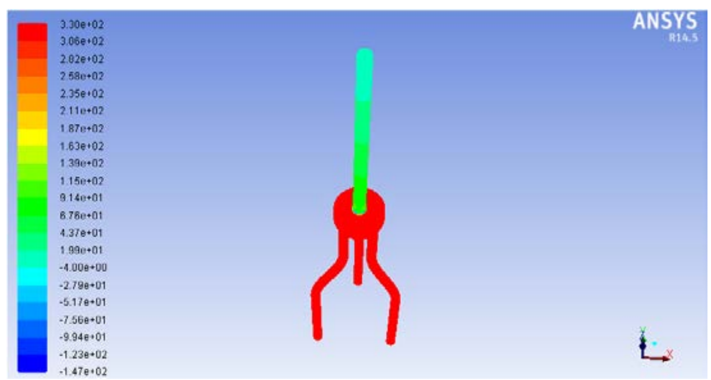
For two intake pipe, pressure is a maximum at the inlet as shown in figure.

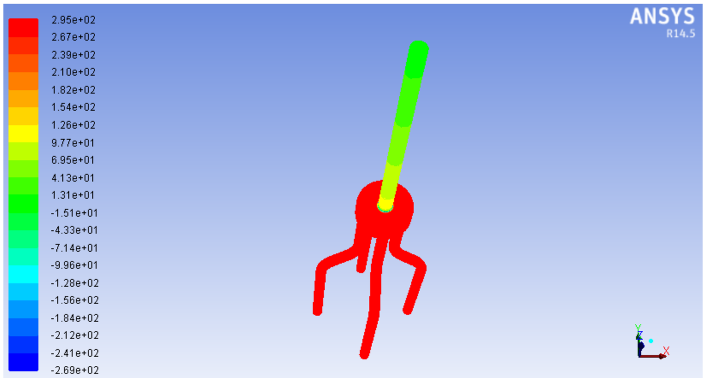
For three intake pipes pressure is maximum at the inlet.

For four intake pipes pressure is maximum at inlet.









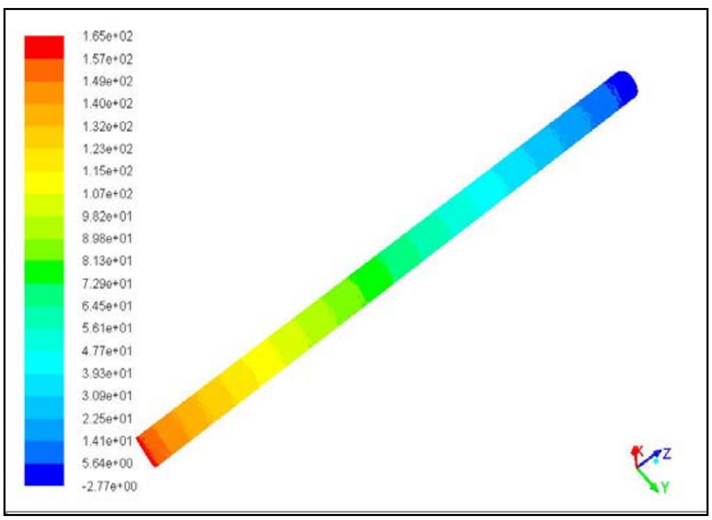
B. Velocity Distribution

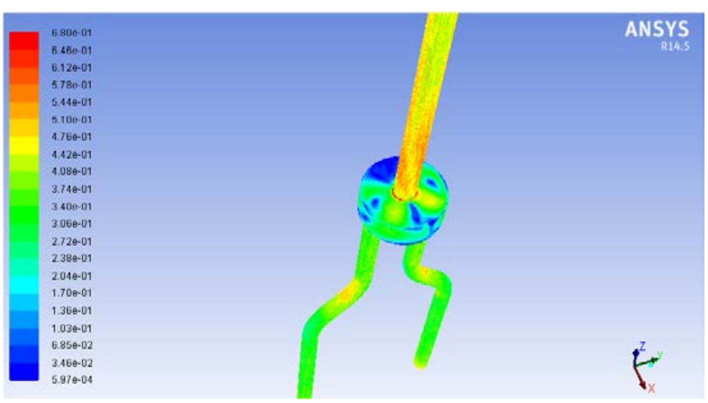
Stream lines calculated by the CFD Analysis are shown in figure. Velocity distributions are shown in Figure. Velocity of the flow is classified by colors. For single pipe, velocity is maximum at centre of pipe and minimum at inner surface of the pipe as shown in fig.

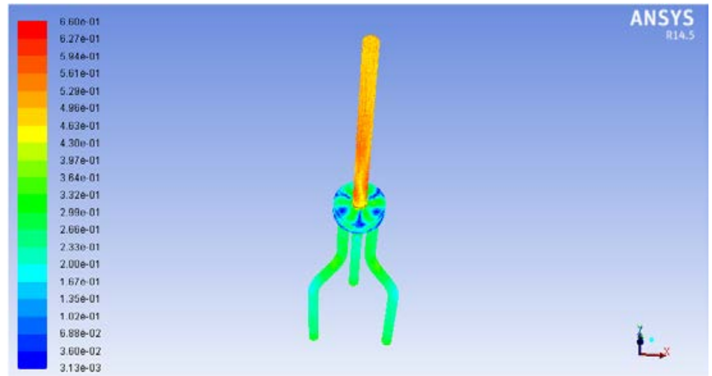
For two intake pipe, a velocity is a maximum in single pipe connected to the two intake manifold while it's minimum in two intake manifold as shown in figure.

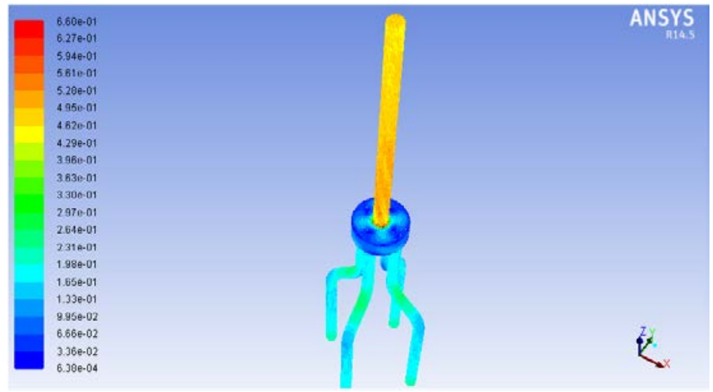
For 3 intake velocity is near about constant as shown in figure.

For 4 intake pipes velocity is minimum at inlet manifold and maximum at outlet single pipe as shown in fig.









VI. CONCLUSION

In the present work enhancement of performance of centrifugal pump through CFD while modifying the suction side for easting discharge has been carried out with boundary conditions such as mass inlet and pressure outlet. Mesh is created using Gambit 2.2.30. Discharge and pressure drop is calculated for three intake pipe.

This study focused on which geometry is best for performance of centrifugal pump through CFD while modifying the suction side for easting discharge. whether single pipe or two intake manifold or three intake manifold or four intake manifold is best geometry for enhancing the discharge by computationally. After selection of geometry we calculate discharge and pressure drop by experimentally.

- Discharge of three intake manifold is high as compared to other geometry.
- Power required for three intake system is less by analytically So, cost is less.
- Pressure drop is less for three intake system by analytically and software analysis.
- For three intake geometry weight is less.
- The best geometry is 3 intake pipe manifold.
- The study confirms that the flow within the pump sump is greatly affected by the geometry of the sump.
- CFD is very helpful in analyzing the effects of sump geometry on the flow pattern.
- Commercial software like ANSYS CFX reduce the time of analysis significantly.

The commercial CFD package ANSYS FLUENT V6 was used to predict the three dimensional flow and vortices in a pump sump model. The CFD model predicts the flow pattern in detail and the location, and nature of the vortices. However, considerable post-processing of the basic data is needed to fully comprehend the details of the flow. Thus CFD model can be used to study the effect of various parameters which reduces time as well as cost and hence can become an important tool for optimization of pump sump geometry.

ACKNOWLEDGMENT

I express my sense of gratitude towards my project guide **Prof. S.V.Channapatanna** for his valuable guidance at every step of study of this dissertation, also his contribution for the solution of every problem at each stage.

I am thankful to **Prof**. **A.A Bagade** Head of the department of Mechanical Engineering and all the staff members who extended the preparatory steps of this dissertation. I am very much thankful to respected Principal **Dr. A. M. Nagraj** for his support and providing all facilities to complete the project.

Finally I want to thank to all of my friends for their support & suggestions. Last but not the least I want to express thanks to my family for giving me support and confidence at each and every stage of this project.

REFERENCES

[1] P.S. Mahar, R.P. Singh (2013), Optimal design of pumping main considering pump characteristics, American society of civil engineering, pp 1949-1204.

[2] Tanweer S. Desmukh, V.K Gahlot (2011), Numerical Study of Flow Behaviour in a Multiple Intake Pump Sump, International Journal of Advanced Engineering Technology, Volume II, Issue II, April-June, pp 118- 128.

[3] Jong-Woong Choi, Young-Do Choi , Chang-Goo Kim and Young-Ho Lee, “Flow uniformity in a multi-intake pump sump model”, Journal of Mechanical Science and Technology, Volume 24, 2010,pp 1389-1400.

[4] A. Rossetti, G. Pavesi, G. Ardizzon, A new two stage miniature pump: Design, experimental characterization and numerical analyses, Sensors and Actuators A, Volume 164, pp 74-87 (2010).

[5] Weng-Guang Li (2000), Effects of viscosity of fluids on centrifugal pump performance and flow pattern in the impeller, International Journal of Heat and Fluid Flow, Volume 21, pp 207-212.

AUTHORS

First Author – Miss vibha pode, B.E., pode.vibha@gmail.com.
Second Author – Prof. S.V.Channapattana, PhD, Rajashri Shahu College of Engineering, Tathavade and svchanna@yahoo.co.in
Third Author – Prof. A.A.Bagade, M.Tech,D.Y.P.S.O.E.A Ambi