NOVATEUR PUBLICATIONS
International Journal Of Research Publications In Engineering And Technology [IJRPET]
ISSN: 2454-7875
VOLUME 1, ISSUE2, 2015

"CFD ANALYSIS OF THE DIFFERENT GEOMETRIC CONFIGURATIONS AT SUCTION SIDE OF CENTRIFUGAL PUMP"

Satish M.Rajmane[1], Dr.S.P.Kallurkar[2]
Research Scholar, WIT Research Center, Solapur University, Solapur, India[1]
(satishrajmane79@gmail.com)
Principal, Atharva College of Engineering, Mumbai, India[2]
(drkallurkar@yahoo.co.in)

ABSTRACT

There are many factor affect pump operation and working which include their physical arrangement, speed, suction head, exhaust head, and properties of liquid etc. lowered capacity, vibration, reduced efficiency, and cavitations could cause serious trouble like Suction Head available, Excessive suction lift, shallow inlet submergence. Category of connection and arrangement are the suction conditions. 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 and modified configurations are studied using CFD techniques while; the best configuration is discovered from the selected geometries.

I. INTRODUCTION

Suction condition is an important factor of pump system design. The Pressure created by the pump 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 proper design an efficient system, critical consideration should be given to suction piping and suction source.

Further, every unit of power saved in the application contributes towards conservation for the environment and towards Green Earth. Every inch of enhancement in the `head' at the output side add to the efficiency of the pumping system. This work aims at improving the performance of the system with a focus on the suction side while contributing to the global effort in upgrading the performance. This work is relevant in the context of lowering power consumption or improving the effective head of the pumping system.

II. LITERATURE REVIEW

In this section a review of research work in the area of different inserts was carried out and based on this review certain observation were made;

Vibha P.Pode, Shylesha Channapattanna[2014] have proposed a study to be done on the suction side of a centrifugal pump, the objective of the study was to improve the performance and provide the best alternative design for the suction side.

Sumit N.Gavande, Prashant D.Deshmukh, Swapnil S.Kulkarni[2014] have studied the various methodologies to increase the discharge of the pump. Some methodologies relate to the change in design of the suction side and some relate to change in design of pump.

Bin Cheng al.[2012] the objective of this study was to analyze the inlet flow characteristics of the lateral diversion and intake pumping stations and access the capacity of flow adjustment of the guide splitter with numerical simulations, based on turbulent model and SIMPLEC algorithm. The main conclusions were, the inlet flow pattern is more complex than single lateral division or lateral intake pumping station and the flow pattern in the lateral diversion part is similar with bend flow;

Honggeng Zhu al.[2012] have investigated the internal flow patterns of a volute type discharge passage, in a mixed flow pumping system based on the Computational Fluid Dynamics(CFD). Analysis shows that the internal flow pattern of volute-type discharge passage is very complex; there is vortex and flow separation in typical cross-sections. Bias flow is obvious in the outlet sections of the volute and the passage, and velocity distribution is not uniform. The distribution uniformity curves of axial velocity in the outlet sections of the volute and the passage are protruding ones opening upward, and the bias angle curves in these two sections are concave ones opening downward. These uniformity curves and bias angle curves reach their maximum or minimum respectively corresponding to the best efficiency point of the pumping system.

HSIAO Shih-Chun al.[2011] has been examined the hydrodynamics of a pump sump consisting of a main channel, pump sump, and intake pipe is examined using Truchas, a three-dimensional Navier-Stokes solver, with a Large Eddy Simulation (LES) turbulence model. The numerical results of stream wise velocity profiles and flow patterns are discussed and compared with experimental data, fairly good agreement is obtained. Furthermore, the proposed numerical model includes the effect of fluid viscosity and considers more realistic simulation conditions. Simulation results show that viscosity affects the prediction of flow patterns and that the stream wise velocity can be better captured by including cross flow. The effects of the submergence Froude number on the free surface and stream wise velocity are also examined. The free surface significantly fluctuates at high submergence Froude number flows and the corresponding distribution of stream wise velocity profiles exhibits a trend different from that obtained for low submergence Froude number flows.

WANG Fu-jun al.[2007] have been successfully simulated the three-dimensional turbulent flows generated by an axial-flow pump equipped with an inducer, using the multiple reference frame approach. The effects of angular alignment of inducer and impeller blades and the axial gap between inducer and impeller have been examined. Numerical simulation results imply that the pressure generated is the maximum when the inducer is aligned at an angle of 0ºor 30º with the impeller. The possible reason to explain these maxima is that the hydraulic loss is minimal when the wake from the inducer impinges on the impeller blades. The effect of decreasing the axial gap reduces the head generated. The predicated results of pump head and efficiency show reasonably good agreement with the experimental data.

GUO Jia-hong al.[2007] has presented a numerical model for three-dimensional turbulent flow in the sump of the pump station. A reasonable boundary condition for the flow in the sump with multi-intakes, each of which may have different flow rates, has been proposed. The finite volume method was employed to solve the governing equations. The fluid flow in a model sump of the pump station is calculated and compared with the experimental results. The comparison between the numerical and the experimental results shows that they fairly agree with each other. Therefore, the present method can be applied to simulate the flow field in the sump with multiple water intakes effectively, and can be used in the design of the sump.

III. MODELING AND ANALYTICAL CALCULATION

The efficiency of the centrifugal pump can be increased by number of ways such as modifying the geometry of the sump, increasing the diameter of the suction pump, having multiple pumps working in series, etc.
This results in better suction of the working fluid and as a result of it the mass flow rate of the fluid increases which directly increases the efficiency of the pump by reducing the motor HP and hence reducing the operational cost of the centrifugal pump.

A) Modeling:-

The following diagram in Figure 3.1 shows a single suction pipe the model is generated in CATIA V5.

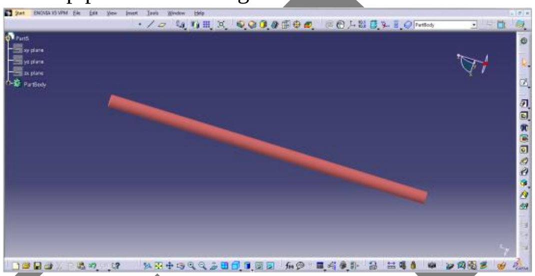

Figure 3.1 Basic Catia Model

Fig 3.2 & 3.3 shows a Modification with two suction pipes the model is in CATIA V5.

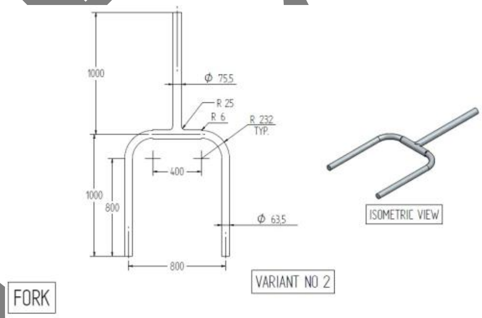

Figure 3.2 first modification Drawing

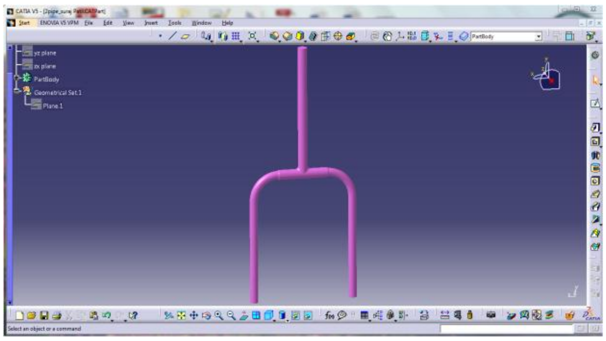

Figure 3.3 First Modification Catia Modle

In Figure 3.4 & 3.5 shows a Modification with three suction pipes the model is generated in CATIA V5.

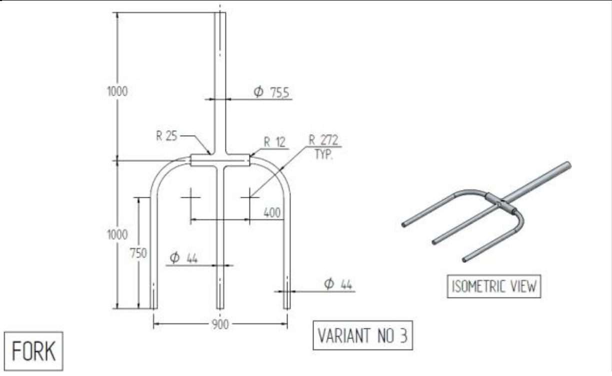

Fig3.4. Second modification Drawing

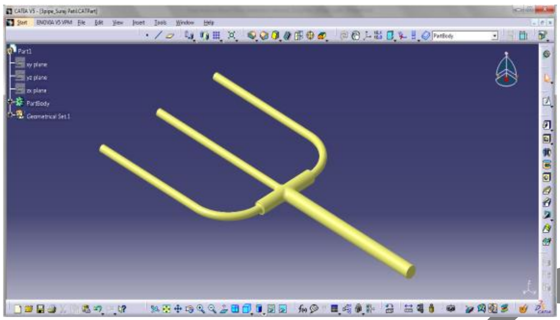

Fig.3.5 Second modification catia model

B) Analytical Calculation

a) Regular suction side. (Single Pipe) :-

Given Data:-Inner diameter of pipe (d_i) = 76.2mm

Thickness of pipe (t) = 3.66mm

$$\therefore Internal Diameter d_i = d_0 - 2t$$

$$\therefore d_o = 76.2 + 2 \times 3.66$$

$$\therefore d_o = 83.52\ mm$$

Length of pipe (l) = 3m

Mass of flowing water per second (m) = 2.5 kg/sec

Density of flowing water (ρ_w) = 1000 kg/m^3

Weight Density of carbon steel (ρ_{cs}) = 7850 kg/m^3

Coefficient of friction for pipe (μ) = 0.005

Efficiency of motor (η) = 80%

I. Cross-sectional area of pipe:

$$a = \frac{\pi}{4} \times {d_i}^2$$

$$\therefore a = \frac{\pi}{4} \times (76.2 \times 10^{-3})^2$$

$$\therefore a = 0.00456\ m^2$$

II. Velocity of flowing liquid in pipe per second:

$$mass = density of water \times area \times velocity$$

$$\therefore velocity, v = \frac{mass}{area \times density of water}$$

$$\therefore v = \frac{2.5}{0.00456 \times 1000}$$

$$\therefore v = 0.548\ m/sec$$

III. Major losses in pipe :

$$h_f = \frac{4flv^2}{2gd_o}$$

$$\therefore h_f = \frac{4 \times 0.005 \times 3 \times 0.548^2}{2 \times 9.81 \times 76.2 \times 10^{-3}}$$

$$\therefore h_f = 0.01206\ m$$

IV. Pressure developed in a pipe

$$p_d = \rho_w g h_f$$

$$\therefore p_d = 1000 \times 9.81 \times 0.01206$$

$$\therefore p_d = 123.606\ N/m^2$$

V. Weight of single pipe:

$$w = specific weight \times volume$$

$$\therefore w = specific\ weight \times area \times length$$

$$\therefore w = \rho_{cs} \times \frac{\pi}{4}({d_0}^2 - {d_i}^2) \times l$$

$$\therefore w = 7850 \times \frac{\pi}{4}[(83.52 \times 10^{-3})^2 - (76.2 \times 10^{-3})^2] \times 3$$

$$\therefore w = 21.614\ kg$$

VI. Power required to pump water:

$$p = \frac{\rho_w Q g h_f}{\eta}$$

$$\therefore p = \frac{1000 \times (2.5/1000) \times 9.81 \times 0.01206}{0.8}$$

$$p = 0.3697 kW$$

Like this for all the modifications the values for pressure drop and power requirement can be calculated which are represented in calculation summary as follows

Calculation Summary:

1. Basic model

Table 3.1 Pressure Drop & Power required for Basic Model

Di	Thickness	Length	Csarea	Velocity	mass	ff	Dcoeff	D Pr. Drop (Bend)	delp	Power required
Mm	mm	M	m2	m/s	kg/s				Pa	Watts
76.2	3.66	3	0.0046	0.5482	2.5	0.005	0	0	118.317	0.36974

2. First Modification (two pipes)

Table 3.2 Pressure Drop & Power required for First Modification

Di	Thickness	Length	Csarea	Velocity	mass	ff	Dcoeff	D Pr. Drop (Bend)	delp	Power required
Mm	Mm	M	m2	m/s	kg/s			Pa	Pa	Watts
76.2	3.66	1	0.0046	0.5482	2.5	0.005			39.439	
63.5	3.4	1	0.0032	0.39471	1.25	0.005	0.2	15.579	40.1134	
63.5	3.4	1	0.0032	0.39471	1.25	0.005	0.2	15.579	40.1134	
									79.5523	0.2486

3. Second Modification (three pipes)

Table 3.3 Pressure Drop & Power required for Second Modification

Di	Thickness	Len.	Csarea	Velocity	mass	ff	Dcoeff.	D Pr. Drop (Bend)	delp	Power required
Mm	mm	m	m2	m/s	kg/s			Pa	Pa	Watts
76.2	3.66	1	0.0046	0.5482	2.5	0.005			39.439	
44	3.25	1	0.0015	0.54805	0.833	0.005	0.2	30.036	98.3008	
44	3.25	1	0.0015	0.54805	0.833	0.005	0.2	30.036	98.3008	
44	3.25	1	0.0015	0.54805	0.833	0.005	0.2	30.036	98.3008	0.430437
									137.74	

IV. FINITE ELEMENT ANALYSIS

We briefly examine the function of each of these elements within the context of a CFD code. Before going for those processes we have to identify the problem; in that first includes define area means exactly for which result we looking for (i.e. pressure drop, mass flow rate, temperature drop etc.). Second things come that is identification of domain in which exact part of complete system have to identify by which expected result (i.e. pressure drop, mass flow rate, temperature drop etc.) will affect and try to examine that part only instead of complete system because it takes long time for simulation and required large memory space in computer system unnecessary.

For solving we are using ANSYS Fluent Solver. In this interface following parameters used:

- System type: Pressure Based system
- Analysis type: Steady state condition
- Model: k-epsilon (2 equations) with realizable model
- Fluid Used: Water
- Mass flow rate at inlet: 2.5 kg/s
- Temperature: 27 deg^{0}C

Post-Processing:

ANSYS Fluent interface used for post –processing.

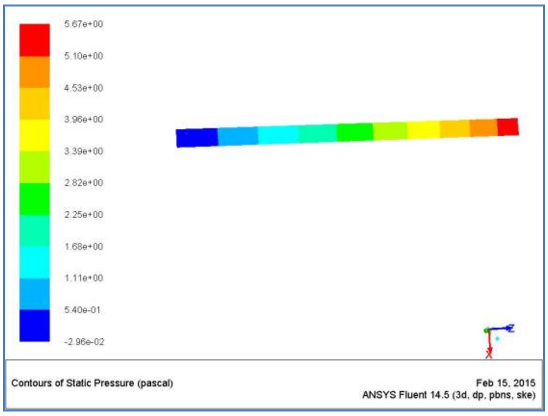

Figure 4.1 Pressure Plot of One Pipe Configuration

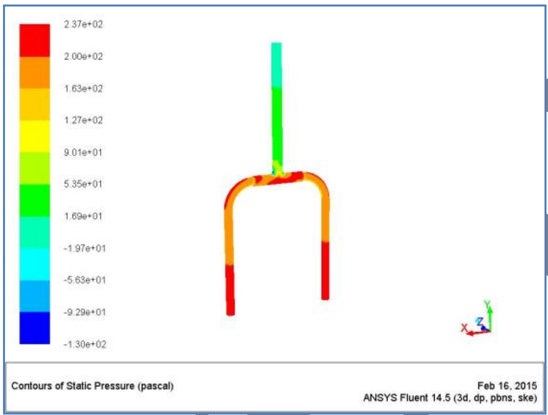

Figure 4.2 Pressure Plot of Two Pipe Configuration

Figure 4.1, 4.2 & 4.3 shows that pressure plot in Pascal. At inlet is more and decreases as gradually along the length. Color strip shows indicate the different pressure level. Blue color indicates the minimum pressure level and red color indicate the max pressure level.

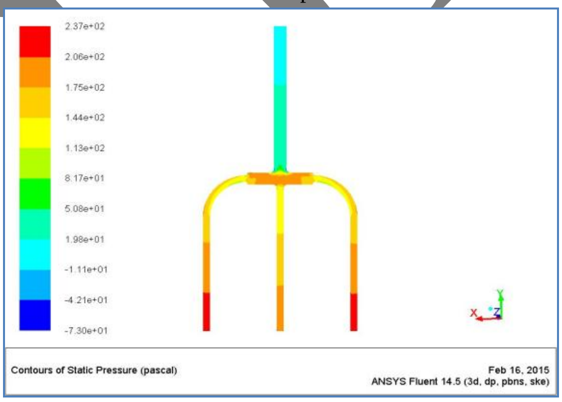

Figure 4.3 Pressure Plot of Three Pipe Configuration

Velocity Plot:

Figures 4.4, 4.5 & 4.6 shows the velocity plot color strip shows the different velocity levels in geometry. At wall o pipe velocity was found minimum and at the centre of pipe velocity was found maximum. Velocity vector shows the flow pattern of fluid. Vortices can be seen using this plot.

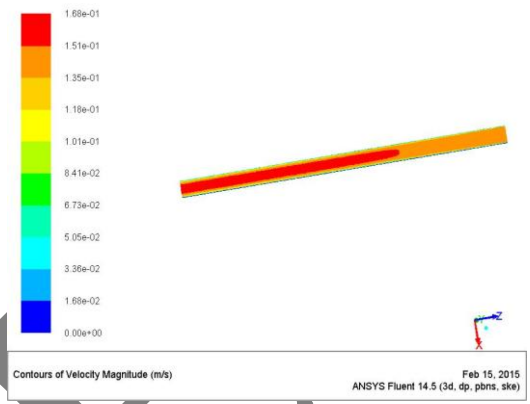

Figure 4.4 Velocity Plot of One Pipe Configuration

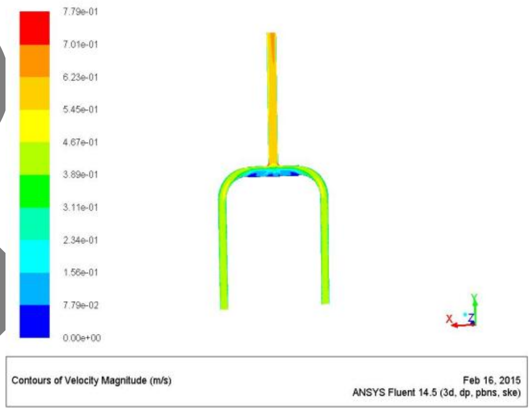

Figure 4.5 Velocity Plot of Two Pipe Configuration.

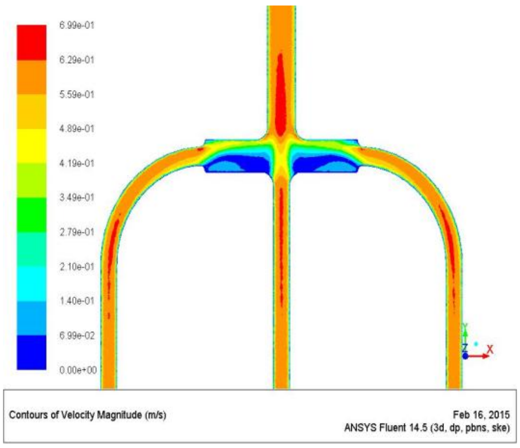

Figure 4.5 Velocity Plot of Three Pipe Configuration

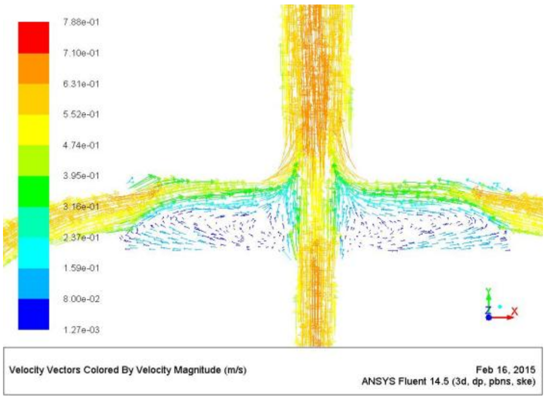

Figure 4.6 Vector Plot of Three Pipe Configuration

V.RESULT

The comparison between single and modified model of suction pipe proposed in our thesis can be very well represented in a tabular format, as shown

Table 5.1 comparison between single and modified model of suction pipe

Sr.No	Diameter	Length	Velocity	Pressure developed (Analytical) in Pascal	Pressure developed (FEA) in Pascal	Power
1	76.2	3	0.548201	118.317	117.3169	0.36974
2	76.2	1	0.548201	79.5523	79.55235	0.248601
	63.5	1	0.394705			
	63.5	1	0.394705			
3	76.2	1	0.657842	137.74	137.7397	0.430437
	44	1	0.548201			
	44	1	0.548054			
	44	1	0.548054			

From above table pressure drop is minimum in 2 pipe configuration & minimum power required for this configuration. Also the Results from mathematical and FEA analysis are nearly same.

Pressure developed in three pipe configuration is more as compared to other two variants. By changing the geometry of the three suction pipes, pressure drop and vortices get minimized. Efficiency of this configuration is more as compared to other variants.

VI. CONCLUSION

- Vortices and cavitations' introduce inefficiency on the operation of the centrifugal pump.
- The suction head and the delivery head has a bearing on the output of the pump in terms of discharge achieved per KW of pump power.
- The intake pumping stations requires a desirable intake flow pattern in order to ensure the operation of pump units.
- The intake pumping stations needs a uniform flow distribution of the sumps in order to ensure the operation of pump units.
- CFD model used to study the effect of various parameters which reduces time as well as cost and hence could become an important tool for optimization of pump sump geometry.
- Redesign of the suction side of the pump facilitated the flow of water and improves the discharge and consequently the performance of the centrifugal pump.

VII. REFRENCES

[1]. Vibha P.Pode, Shylesha Channapattanna, Evaluating Performance Of Centrifugal Pump Through Cfd While Modifying The Suction Side For Easting Discharge, International Journal Of Research In Engineering And Technology Eissn: 2319-1163 | Pissn: 2321-7308, Jan-2014

[2]. Sumit N.Gavande, Prashant D.Deshmukh, Swapnil S.Kulkarni, A Technique To Enhance The Discharge Of A Multi Intake Centrifugal Pump, International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974, Volume: 03 Issue: 01 | Jan-2014

[3]. S. Bin Cheng, Yonghai Yu, CFD Simulation and Optimization for Lateral Diversion and Intake Pumping Stations, 2012 International Conference on Modern Hydraulic Engineering, Procedia Engineering 28 (2012) 122 – 127.

[4]. Honggeng Zhu, Rentian Zhang, Guoqiang Luo, Bin Zhang, Investigation of Hydraulic Characteristics of a Volute-type Discharge Passage based on CFD, 2012 International Conference on Modern Hydraulic Engineering, Procedia Engineering 28 (2012) 27 – 32

[5]. CHUANG Wei-Liang, HSIAO Shih-Chun, Three-Dimensional Numerical Simulation Of Intake Model With Cross Flow, Journal of Hydrodynamics 2011,23(3):314-324, DOI: 10.1016/S1001-6058(10) 60118-7.

[6]. Li Yao-jun, Wang Fu-jun, Numerical Investigation Of Performance Of An Axial-Flow Pump With Inducer, journal of Hydrodynamics, Ser.B, 2007,19(6):705-711

[7]. Chen Hong-xun, GUO Jia-hong, Numerical Simulation Of 3D Turbulent Flow In The Multi intakes Sump Of The Pump Station, journal of Hydrodynamics, Ser. B, 2007,19(1):42-47