Improved Performance of Polypropylene Centrifugal Pump Using CFD

Gautam Nain[1], Rahul Malik[2]
[1] M.Tech Scholar, Department of Mechanical Engineering, PMCE, Sonipat
[2] Head of Department, , Department of Mechanical Engineering, PMCE, Sonipat

ABSTRACT

The suction side of the pump typically consists of a single pipe of a nominal bore same as or greater than the delivery pipe. This arrangement may not be effective for higher capacity pumps used in the industry or public distribution system. The effort of this dissertation work would be to identify suitable configuration for the suction side of the Centrifugal Pump to enhance the utilization of the power; in other words, to maximize the discharge to power ratio. Quantitative techniques using CFD would be utilized to solve the problem for assessing the amount of discharge realized by a given configuration of the system with single or multiple channels for intake at the suction side. Mathematical treatment would be offered for addressing the preliminary investigation into the problem. ANSYS Fluent would be used to evaluate the variants for the intake manifold for determining the discharge and the pattern of flow of water through the manifold. Minimal pressure drop combined with a high value of discharge would be the criteria for assessment of the variants.

Keywords: Pump, suction, optimization.

1. Introduction

The centrifugal pump is a member of family known as rotary machine consists of two basic parts 1.The rotary element or impeller.2.The stationary element or casing (volute). A centrifugal pump delivers useful energy to the fluid on pump age largely through velocity changes that occur as this fluid flows through the impeller and the associated fixed passage ways of the pump. It is converting of mechanical energy to hydraulic energy of the handling fluid to get it to a required place or height by the centrifugal force of the impeller blade. The input power of centrifugal pump is the mechanical energy and such as electrical motor of the drive shaft driven by the prime mover or small engine. The output energy is hydraulic energy of the fluid being raised or carried. In a centrifugal pump, the liquid is forced by atmospheric or other pressure into a set of rotating vanes. A centrifugal pump consists of a set of rotation vanes enclosed within a housing or casing that is used to impart energy to a fluid through centrifugal force. A pump transfer mechanical energy from some external source to the liquid flowing through it and losses occur in any energy conversion process. The energy transferred is predicted by *the Euler Equation*. The energy transfer quantities are losses between fluid power and mechanical power of the impeller or runner. Thus, centrifugal pump may be taken losses of energy. The performance of pumps depends upon number of hydraulic considerations such as net positive suction head, suction flow conditions and operating range in terms of flow and head. Though net positive suction head are well understood and adequately addressed in the design of pumping stations, the sizing and arrangements of suction manifold is based on good industry practice guidelines. Poor flow conditions in suction manifolds can lead to non-uniform distribution of flow across the pipe cross-section and swirling, resulting in pump vibration. Due emphasis needs to be paid on the bends, tees and configuration of manifolds on flow imbalance and swirling.

The rest of paper is design as given as. The overall past work is describe in Section II. Section III describes the methodology used for proposed work. Result analysis describe in section IV. Finally, the conclusion of paper is described in section V.

2. Literature Review

The task deals with the optimization and analysis in suction of centrifugal pump so in related to this following research papers have been studied:-

Cheng et al. analyzed 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. The main conclusions are as follows:(1)

the numerical simulation of the pumping station suggests that the inlet flow pattern is more complex than single lateral division or lateral intake pumping station [1].

Zhuet al. had found that the 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 out let sections of the volute and the passage, and velocity distribution is not uniform. [2].

WANG et al. concluded that three-dimensional turbulent flows generated by an axial-flow pump equipped with an inducer have been successfully simulated 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 inducers aligned at an angle of 0o or 30o with the impeller [3].

GUO et al. designed a numerical model for three-dimensional turbulent flow in the sump of the pump station has been presented. A reasonable boundary condition for the flow in the sump with multi-intakes, each of which may have different flow rates, has been proposed[4].

Asok et al. had found three-dimensional CFD analyses have been demonstrated in the prediction of pressure drop taking place in helical-grooved labyrinth seals having good agreement with experimental results. Helical-grooved labyrinth seals have better pressure reduction characteristics over the circular-grooved and/or sinusoidal grooved seals [5].

HSIAO et.al studied 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. [6].

S. C. Chaudhari, C. O. Yadav & A. B. Damor described an improve the head of mixed flow pump impeller, Computational Fluid Dynamics (CFD) analysis is one of the advanced CAE tools used in the pump industry. From the results of CFD analysis, the velocity and pressure in the outlet of the impeller is predicted[7].

Kapil Pandya1, Chetankumar M.Patel has presented various approaches used in CFD analysis of centrifugal pump and highlights the advantages and application of CFD analysis in turbo industries. The CFD analysis is the advanced tool to overcome the limitation of conventional method to design the pump[9].

K.W Cheah, T.S. Lee & S.H Wino to investigated within a centrifugal pump with six twisted blade impeller is carried out to understand the impeller and volute tongue interactions. The numerical analysis is done by solving the three-dimensional RANS codes with standard k-e two-equation turbulence model. Wall regions of the computational domain are modeled with a scalable log-law wall function. [10].

P.Gurupranesh, R.C.Radha, N.Karthikeyan enhanced the performance of the centrifugal pump through design modification of impeller. Theories on pump characteristics are studied in detail. Vane profile of the impeller are generated using point by point method [11].

3. METHODOLOGY

The present work has been followed the certain methodology which provided the optimum approach to finish the work. For CFD analysis of pump *first step* to generate CAD model which has been done be CATIA V5 software, this stage is pre-processor stage for the simulation *Second step* to assemble the parts and generate the mesh different pump components with the help of ANSYS software, which is also a pre processer stage for simulation. *Third step* is solving the equation with help of boundary condition and initial condition with the help of FLUENT software, this stage is called solver stage.

Final step analyze the result, this stage is called post processing stage. The methodology of CFD analysis has been shown by flow chart given below in figure 1

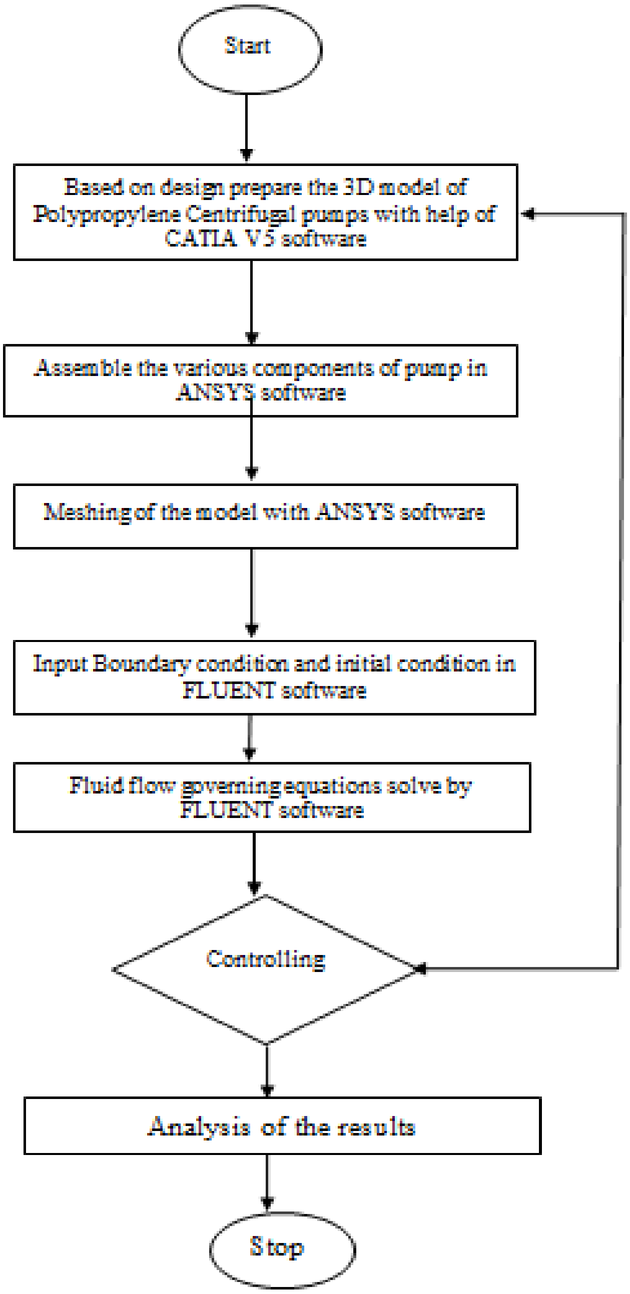


Figure 1: Flow chart for methodology

4. Result Analysis

In this paper we will discuss how to design the suction side. 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.

Fig 2 Maximum Total Pressure 137.7397 Pa. on impeller

The variation of pressure in the impeller is shown below in figure 3. It has been found that the upper portion of impeller have maximum value

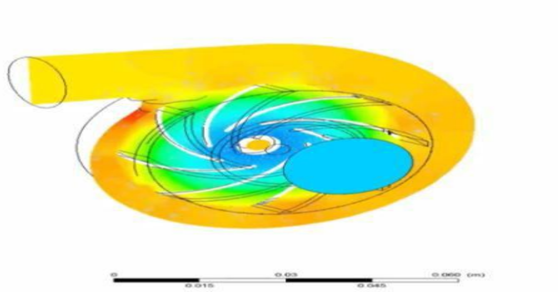


Fig 3 Maximum Total Pressure 137.7397 Pa. on impeller

Fig 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. The variation of total pressure in tubular casing with draft tube is shown below in figure 4. It has been found maximum value of pressure obtained at inlet of the casing and the total pressure has minimum value at the inlet of draft tube.

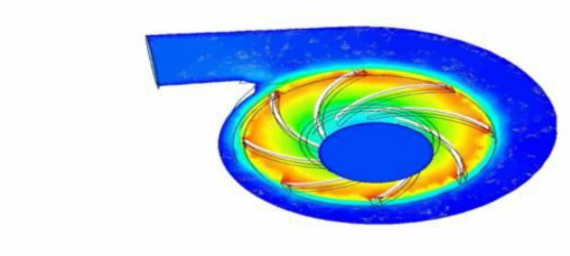


Fig 4 Velocity contour in Pump 0.548054 m/s

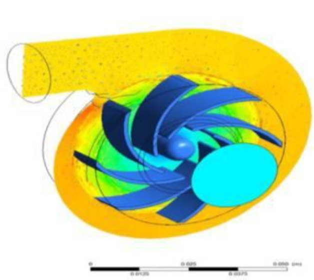
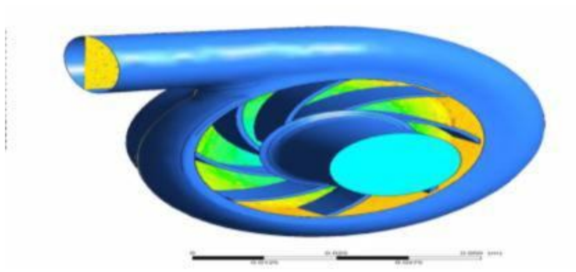


Fig 5 Velocity vector in Pump 0.548054 m/s



Velocity Plot:
Velocity plot shown in fig 7. color strip shows the different velocity levels in geometry. At wall on pipe velocity is minimum and at the centre of pipe velocity is max. Velocity vector shows the flow pattern of fluid. Vortices can be seen using this plot.

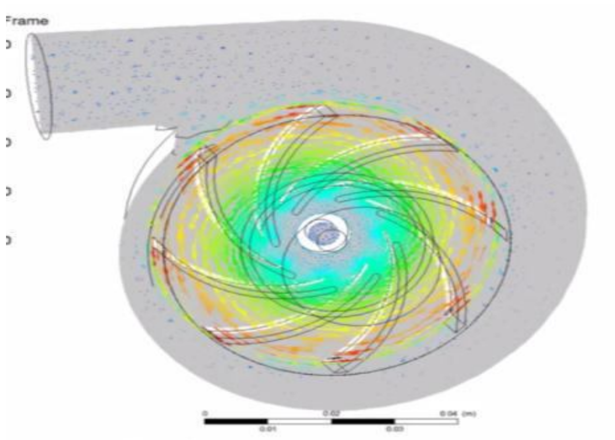


Fig 7 Velocity vector at Pump plane 0.548054 m/s

Graph Plot:

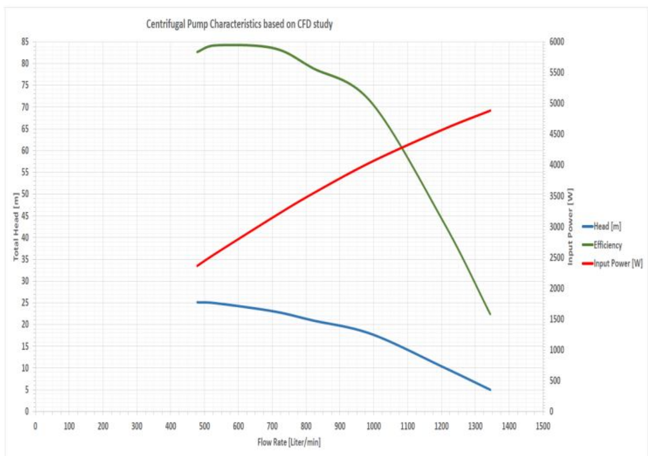


Fig 8 Graph shown in fig. centrifugal pump characteristics based on CFD.

Pressure generated due to change in dia. Size

The pressure will increase due to change in diameter size. The figure 5.8 shows the variation of the pressure with respect to diameter change

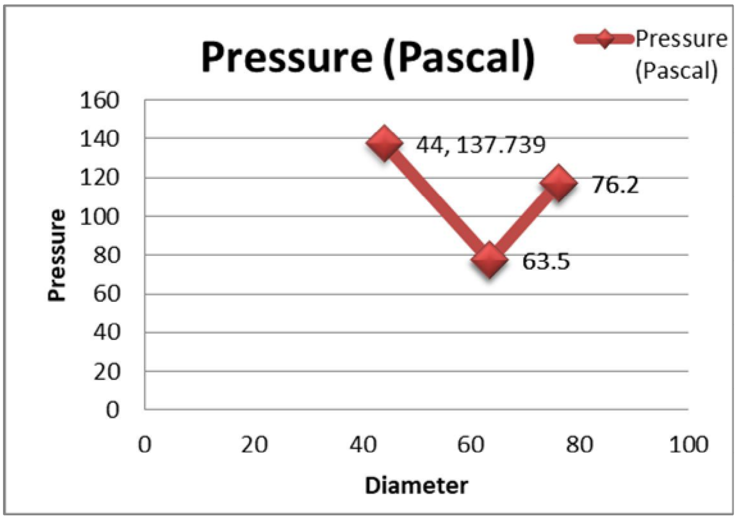


Fig 5.15 Pressure v/s Diameter

Power generated due to change in dia. Size with respect to length

The power will increase due to change in diameter size. The figure shows the variation of the power with respect to diameter change. The power will be 0.43043 KW for the diameter of 44mm.

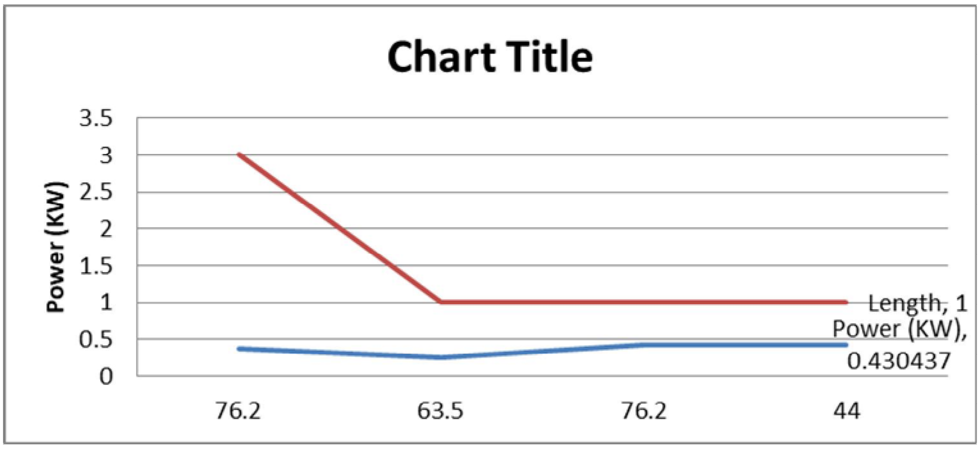


Fig 5.15 Diameter v/s Power v/s Length

5. Conclusion

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 below From below table pressure drop is minimum in 2 pipe configuration. Also minimum power required for this configuration. 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.

References

[1] 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), 27-32

[2] 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), 122 – 127
[3] 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
[4] CHEN Hong-xun, GUO Jia-hong, Numerical Simulation Of 3-D Turbulent Flow In The Multiintakes Sump Of The Pump Station, Journal of Hydrodynamics Ser.B, 2007,19(1):42-47
[5] S.P. Asok, K. Sankaranarayanasamy, T. Sundararajan, G. Vaidyanathan, K. Udhaya Kumar, Pressure drop and cavitation investigations on static helical-grooved square,triangular and curved cavity liquid labyrinth seals, Nuclear Engineering and Design 241 (2011) 843– 853
[6] 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
[7] S. C. Chaudhari, C. O. Yadav & A. B. Damor, A comparative study of mix flow pump impeller cfd analysis and experimental data of submersible pump, International Journal of Research in Engineering & Technology (IJRET) ISSN 2321-8843 Vol. 1, Issue 3, Aug 2013, 57-64.
[8] kapil Pandya, Chetankumar M.Patel, A Critical review on CFD Analysis of centrifugal pump impeller, International Journal of Advance Engineering and Research Development (IJAERD) Volume 1,Issue 6,June 2014, e-ISSN: 2348 - 4470 , print-ISSN:2348-6406
[9] K.W Cheah, T.S. Lee, and S.H Winoto, Unsteady Fluid Flow Study in a Centrifugal Pump By CFD Method, 7th ASEAN ANSYS Conference Biopolis, Singapore 30th and 31st October 2008
[10] P.Gurupranesh, R.C.Radha, N.Karthikeyan, CFD Analysis of centrifugal pump impeller for performance enhancement, IOSR Journal of Mechanical and Civil Engineering (IOSR JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X PP 33-41
[11] A. E. A. Aqeel, Technology Engineering Software Limited CFX TASC flow Computational Fluid Dynamics Software: Primer Manual, Version 2.10 (2000).
[12] J. Wang, Piechnajanusz et al. A novel design of composite water turbine using CFD, J. Hydrodynamics, 24(1), 11-16 (2012).
[13] K. Srinivasan. Rotodynamic pumps (centrifugal and axial), New age international (2008).
[14] R. I. Lewis, Turbomachinery Performance Analysis, John Wiley, New York (1996).
[15] J. M. Lighthill, A new method of two - dimensional aerodynamic design (1945).
[16] S. Rajendran and K. Purushothaman, Analysis of a centrifugal pump impeller using ANSYS-CFX, Int. J. Engg. Res. Technol., 1(3) (2012).
[17] M. B. Flathers and G. E. Bache, Aerodynamically induced radial forces in a centrifugal gas compressor, J. Engg. Gas Turbines & Power (1999).
[18] R. K. Byskov and C. B. Jacobsen, Flow in a centrifugal pump impeller at design and off-design conditions-Part II: Large eddy simulations, J. Fluids Engg. (2003).
[19] W. N. Dawes, P. C. Dhanasekaran, W. P. Kellar and A. M. Savill, Reducing bottlenecks in the CAD-to-Mesh-to-Solution cycle time to allow CFD to participate in design, J. Turbo Machinery (2001).
[20] S. L. Chen and W. T. Wang, Computer aided manufacturing technologies for centrifugal compressor impellers, J. Mater. Process. Technol. (2001).