International Journal of Scientific Research and Management Studies (IJSRMS)
ISSN: 23493771 Volume 2 Issue 1, pg: 52-58

QUANTITATIVE ANALYSIS OF VARIANTS FOR SUCTION NETWORK AND ITS EFFECT ON POWER REQUIREMENT OF CENTRIFUGAL PUMP

[1]Satish M. Rajmane, [2]S.P.Kallurkar, [3]Swapnil S. Kulkarni.
[1]Research Scholar for WIT Research Center, Solapur University, Solapur, India
[2]Principal, Atharva College of Engineering, Mumbai, India
[3]Director – Able Technologies (I) Pvt. Ltd., Pune, India

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 multi–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— *Quantitative Analysis, CFD, ANSYS Fluent, Flow through pipe, Discharge for centrifugal pump*

I. INTRODUCTION

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.

For a typical pumping station with multiple pumps, the suction pipe work comprises a manifold located either inside or outside the pumping station. From which individual suction lines run to the pumps. The individual suction lines include and isolation valve, a taper ad one or more bends. The angle of the suction lines to the manifold could vary between 90 degrees for small pumping stations to 45 degrees for larger pumping stations. All of these tees, bends, tapers and valves have an influence on the flow as it travels to the pumps.

"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 efficacy 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. "

Problem Statement:

Hydraulic design of a pumping station is critical for achieving optimal performance of pumps. The performance of pumps depends upon a number of hydraulic consideration such as net positive suction head, suction flow consideration and operating range in term of flow condition. Poor flow conditions in suction manifolds can lead to non uniform distribution of flow across the pipe cross section. Improve the flow in pipes by reducing the flow separation in the bending and sharp areas by changing the geometry. The discharge of the pump is severely affected while compared with its theoretical discharge possible for that size of pump. The use of the electrical power, as a result, is inefficient system for such purposes of lifting water in a massive volume. Consequently, the time and power required to deliver the output too is higher.

Aim:

To improve the efficiency of the pump by considering redesign of the peripherals either to the suction or the delivery side

Objectives:

- Identify the problem areas by studying the existing flow distribution systems
- Document the challenges to be addressed for enhancing the effectiveness of the suction side flow distribution
- Consider feasibility for redesign of the suction side of the Pump such as tapered section and bends etc.
- Analyze the multi-intake manifold design using CAE software, especially in the CFD domain
- Recommend the best alternative design for the suction side of the pumping system from variants such as combination of two or more pipes and its different arrangements.
- Validate using alternative methodology (physical experimentation)

II. LITERATURE REVIEW

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

1) Cheng et al. Study was undertaken 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. 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 and the flow pattern in the lateral diversion part is similar with bend flow.(2) The lateral diversion and intake pumping stations require not only a desirable intake flow pattern but also need a uniform flow distribution of the sumps in order to ensure the operation of pump units.(3) In the inlet section of the lateral intake part of this type of pumping stations, the guide splitter has an obvious adjustment of the momentum distribution which related to the velocity distributions of the inlet section of the sumps.(4) The numerical simulation results denote that the reverse flow in the lateral diversion part is weaken which has negative effect of normal ranger vertical velocity and the flow uniformity of the sumps is increased with a suitable guide splitter arrangement.

2) 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. 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 down ward. These uniformity curves and bias angle curves reach their maximum or minimum respectively corresponding to the best efficiency point of the pumping system. The head loss of volute-type discharge passage varies with the flow rate, which reaches its minimum when the pumping set efficiency is at the peak. When the pumping system operates under off-design conditions and on the large flow rate side, the head loss increases rapidly and the pumping set efficiency decreases sharply, hence, appropriate adjusting methods should be adopted to realize economic operation of pumping station.

3) 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. 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. When the axial gap between the inducer and impellers reduced to 3 percent's of the impeller diameter, the head coefficient of the pump drops by about 0.02.This can be attributed to interference between inducer and impeller. The predicated results of pump head and efficiency show reasonably good agreement with the experimental data. The axial velocities at the leading and trailing edge of the impeller blade have also been predicted. But, the uniform distribution pattern that is widely used for the design of axial-flow pump is not observed. A non-uniform axial velocity distribution pattern is recommended for the optimal hydraulic design of the axial-flow pump impeller in the future.

4) GUO et al. In his article, 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. 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.

5) 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. They bring in additional energy losses due to flow bending effects and vortices operating in the azimuthal direction also. Regression analysis based genetic algorithm scheme has been demonstrated as one of the means of optimization of seals. The pressure drop ratio of optimal HSLS is 27% more than that of an optimal circular-grooved square cavity labyrinth seal. Curved cavity designs infuse mobility into stagnation pockets existing inside seal cavities. This leads to higher pressure drops because of enhanced dissipation of flow momentum. A particular configuration of HCLS profile whose cavity contains two curved segments intercepted by a small horizontal flat strip, exhibits large internal cavity turbulence and outperforms the other seals. The seals did not show signs of aggressive cavitation during pressure drop studies. CFD cavitation analyses of seals also suggest suitability of the newer seals for the considered application due to their ability to smoothly create high pressure drops without exhibiting high intensity of cavitation.

6) HSIAO et al. was 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. The numerical results of stream wise velocity profiles and flow patterns are discussed and compared with experimental data of Ansar and Nakato. Fairly good agreement is obtained. Furthermore, unlike Ansar et al.'s in viscid solution, 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.

7) S. C. Chaudhari, C. O. Yadav & A. B. Damor has presented " *A comparative study of mix flow pump impeller cfd analysis and experimental data of submersible pump"* which describes 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. The optimum inlet and outlet vane angles are calculated for the existing impeller by using the empirical relations. The CAD models of the mixed flow impeller with optimum inlet and outlet angles are modeled using CAD modeling software Solid Works 2009. By changing the outlet angle and the number of blade of impeller the head of the impeller is improved.

8) Kapil Pandya1, Chetankumar M.Patel has presented *"A Critical review on CFD Analysis of centrifugal pump impeller",* the main objective of which is to go through various approaches used in CFD analysis of centrifugal pump and highlight 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. Now a days CFD analysis is very familiar approach to improve the design of centrifugal pump and optimize its operational parameters like Head, Power, Discharge and Speed. The performance prediction of centrifugal pump using experimental approach cannot fulfill the desired outcome for the researcher. The CFD analysis provides more options towards the evaluation and synthesis of pump.

9)K.W Cheah, T.S. Lee & S.H Winoto *presented "Unsteady Fluid Flow Study in a Centrifugal Pump by CFD Method"* in which investigation 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-equations turbulence model. Wall regions of the computational domain are modeled with a scalable log-law wall function. Current numerical modeling is done with multiple frames of reference and the dissimilar tetrahedral mesh interface of the impeller/volute casing is connected with sliding mesh technique. The analysis shows that there is a recirculation zone near to suction-front shroud side just after the leading edge even at design point. However, the flow within the impeller passage is very smooth and following the curvature of the blade in stream-wise direction. The results of existing analysis proved that the pressure fluctuation periodically is due to the position of impeller blade relative to tongue and the flow field within the volute casing is always unsteady and turbulent.

10) P.Gurupranesh, R.C.Radha, N.Karthikeyan has presented a paper "*CFD Analysis of centrifugal pump impeller for performance enhancement"* In which the project is devoted to enhance 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. The impeller is modelled in Solid works 2012 software and CFD analysis is done using fluid flow simulation package. CFD analysis enables to predict the performance of the pump and a comparative analysis is made for the entire control volume.

III. METHODOLOGY

Mathematical Model:

[1] 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. In the modified Newton-Raphson method we approximate the Jacobian (which really should change in each iteration, by a fixed matrix. Now, we no longer need to re-compute the Jacobian in every iteration, nor do we need to solve the linear system (invert the Jacobian) in every iteration.

Analytical Method

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. Process is carried out in four steps:

Step 1: 3D Modeling

Geometry creation using CAD software such as CATIA, UGNX. For fluid analysis we are creating a fluid domain for a system.

Step 2: Pre-processor

In pre-processor, Gambit software is used. Generally tetrahedral elements are used for meshing. All boundary conditions are defined in this interface.

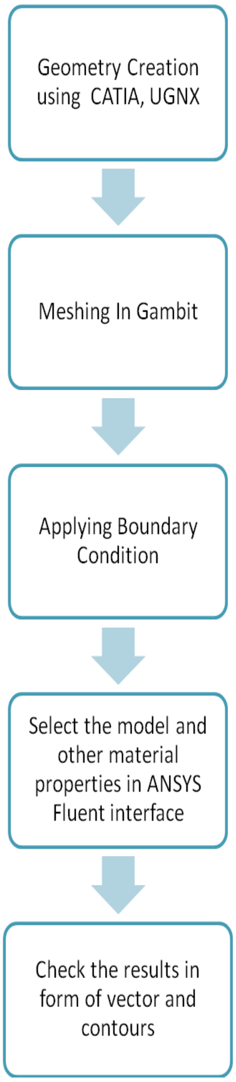
Step 3: Processing

ANSYS fluent workbench is used for processing. In this workbench typically k-epsilon turbulent model with 2 equations is used for solving purpose.

Step 4 : Post-processing

In post processing we are using ANSYS Fluent workbench. We can calculate the velocity and back pressure in the form of contour and vector plot.

General Process Flow chart:



IV. Experimentation And Validation

Time required for a constant volume of discharge varies with geometry. During experimentation, time required for constant discharge and back pressure is determined for the optimum variant. In this process we shall use the 1 HP or ½ HP motor (Pump) and standard pipe size for making prototype. This will be followed with comparison of the results derived by analytical and numerical methodology.

V. Proposed Steps Of The Work

PHASE [1]: Study the design of existing suction pipe.

PHASE [2]: Analysis of the existing suction side pipe.
The analysis of the existing suction side pipe will be done on the current parameters of the suction pipe to determine the discharge.
PHASE [3]: Modification in the geometry of the suction side pipe by introducing suction side manifold.
In this we will replace the single suction pipe with multiple pipes having manifold. The design will consist of two, three and four pipe meeting at the manifold.
PHASE [4]: Analysis of the Modified design alternative.
An Appropriate correction factors need to be considered while making the comparison due to effects of the environment in the physical setup of the experiment. The data captured during experimentation would be compared with the results which are obtained from computational method. If the values are in the permissible limits then a good match would indicate a strong basis for validation of this work.
PHASE [5]: Recommendation of the best suitable alternative
Proper selection of multi intake suction manifold will result in enhancement in terms of discharge. This predictive capability will be very useful in optimizing the operating cost of system without carrying out the costly and tedious trial-and-error processes.

VI. DISCUSSION

The Literature survey helps us to identify various methodologies to increase the discharge by numerous parameters such as modification in design of pump, variation in the design of sump, the geometry of suction and discharge pipe, reducing the vortices, etc. This study helps to explore a new method to increase the discharge. Various design of the suction side manifold can help us to determine the best design to attain more discharge.

VII. CONCLUSION

The data captured during experimentation would be compared with the result for experimentation. A good match or concurrence would indicate a strong basis for validation of the work. Appropriate correction factors needs to be considered while making the comparison due to effects of the environment or the variation in the physical setup or the condition of experiment.

ACKNOWLEDGEMENT

Authors are thankful to Able Technologies (I) Pvt. Ltd., Pune, for sponsoring project work. Authors are also thankful to Mr.S.K.Patil who is ME student & In plant trainee at Able Technologies (I) Pvt. Ltd., Pune.

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

AUTHORS BIOGRAPHY

S.M.Rajmane.is having ME from Solapur University. He is Research Scholar for WIT Research Center, Solapur University, Solapur. He is having more than 10 years teaching experience. His area of interest is in Fluid machinery, FEM, Design engineering.

S.P.Kallurkar is having Phd from NITIE Mumbai .Presently working as Principal in Atharva College of engineering, Mumbai. He is Research Guide for WIT research center, Solapur University, Solapur. He is having more than 25 years teaching experience.

Swapnil S.Kulkarni Director, Able Technologies(I) India Pvt. Ltd., Pune. The Company offers Engineering Services and Manufacturing Solutions to Automotive OEM’s and Tier I and Tier II Companies. He is a Graduate in Industrial Engineering with PG in Operations Management. With around 20 years of working experience in the domain of R&D, Product Design and Tool Engineering, he has executed projects in the Automotive, Medical and Lighting Industry. His area of interest is Research and Development in the Engineering Industry as well as the emerging sector of Renewable Energy

.