A Review Paper on Performance Enhancement of a Centrifugal Pump

Jaydeep G. Chaudhari[1] Mr. A.S. Dandiwala[2] Mr. A. V. Doshi[3]
[1]M.E. Student [2,3]Assistant Professor
[1,2,3]Department of Mechanical Engineering
[1,2]Shree S'ad Vidya Mandal Institute of Technology, Bharuch, Gujarat, India [3]Sardar Vallabhbhai National Institute of Technology, Surat, Gujarat, India

Abstract— The study of paper concerns with the research work done by the various researchers in the field of optimization and analysis of centrifugal pump performance using CFD tool. The use of CFD-tools to analyses the flow field in turbo machines and to predict performance parameters has gained enormous popularity in recent years. Pump is a turbo machines in which mechanical energy is transformed into the hydraulic energy. Various components of centrifugal pump must be designed carefully for the better performance of the centrifugal pump.

Key words: Centrifugal Pump, CFD-tools, CAD models

I. INTRODUCTION

The centrifugal pump is the most used pump in the world. The principle is simple, well described and thoroughly tested. And the pump is robust, effective and relatively inexpensive to produce. There is wide range of variation based on the principle of the centrifugal pump and consisting of the same basic hydraulic parts. Centrifugal pumps are often use in many industrial, municipal and commercial applications and are usefully quick to install and easy to repair.

An increase in the fluid pressure from the pump inlet to its outlet is created when the pump is working. This pressure difference drives the fluid one place to another in the system or plant.

The centrifugal pump creates an increase in the pressure by transferring mechanical energy to fluid through the rotating impeller. The fluid flows from the inlet to the impeller centre and out along its blades. The centrifugal pump increases the fluid velocity and also the kinetic energy is transformed to pressure.

A. Working principle of centrifugal pump

The centrifugal pump acts as a reverse of an inward radial flow reaction turbine. This means that the flow in centrifugal pump is in the radial flow direction. The centrifugal pump works in the principle of force vortex flow which means that the certain mass of liquid is rotate by an external torque the rise in pressure head of rotating liquid take place. The rise in pressure head at any point of the rotting liquid is proportional to the square of tangential velocity of the liquid at that point. Thus at the outlet of the impeller, where radius is more the rise in pressure head will be more and the liquid will be discharged at the outlet with a high pressure head. Due to this pressure head the liquid can be lifted to a high level.

B. Performance of Centrifugal Pump

Pump performance can be predicted by the pump performance curve. Every aspect of performance of centrifugal pump can be predicted by the characteristics curves. Characteristics curves of centrifugal pump can be constructed by the pump performance parameter. Pump performance parameter are of two types (a) Dimensional parameter, they are Flow rate, Head, Power input and rpm and (b) Non-dimensional parameter, they are Head number, Flow number, Power number and Efficiency.

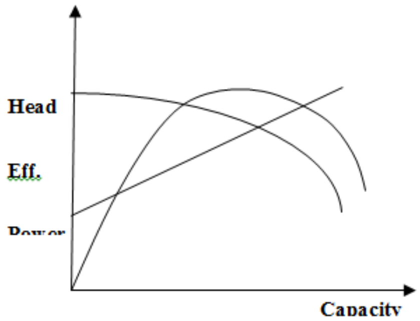

Fig. 1 Performance of centrifugal pump

Flow analysis in centrifugal pump has long been an intensive subject of research. Computational Fluid Dynamics (CFD) analysis is one advanced tool used in the pump industry. This process has been stimulated by the availability of commercial packages allowing flow analysis in a pump. Although it is very well known that the flow field in a centrifugal pump is inherently unsteady, most calculations nowadays are done with steady methods. All this has to do with the cost of unsteady calculations.

Numerical analysis using CFD tools involves geometry definition and grid generation of computational domain. Further this process includes selection of grid types, grid refinements and defining correct boundary conditions. During the course of this work, fine mesh is generated for impeller blade zone to capture complex flow behavior inside blades.

II. LITERATURE SURVEY

A. Manivannan [1] studied to predict the flow pattern inside the impeller. From the results of CFD analysis, the velocity and pressure at the outlet of the impeller ware predicted. CFD analysis was done using Star CCM+ software. Outlet flow conditions were used to calculate the efficiency of the impeller. The calculated value of efficiency from the empirical relations was 55%. The optimum inlet and outlet vane angles were calculated for the existing impeller by using the empirical relations. The CAD models of the mixed flow impeller with optimum inlet and outlet angles were modeled using CAD modeling software Pro-E WF3. To find the relationship between the vane angles and the impeller performance the optimum vane angle was achieved step by step. Three CAD models were generated with the vane angles between existing and optimum values. Those models were analyzed individually to find the performance of the impeller. In the first case, outlet angle was increased by 5°. From the outlet flow conditions, obtained from the CFD analysis, it was evident that the reduced outlet recirculation and flow separation cause the improved efficiency. By changing the outlet angle the efficiency of the impeller was improved to 59%. In the second case inlet angle was decreased by 10%. The efficiency of the impeller in that case was 61%. From that analysis it was understood that the changes in the inlet vane angle was not changing the efficiency of the impeller as much as the changes in outlet angle. In the third case, impeller with optimum vane angles was analyzed and the outlet flow conditions were predicted. From the CFD analysis the efficiency of the impeller with optimum vane angles was calculated as 65%. Thus, efficiency of the mixed flow impeller was improved by 18.18% by changing the inlet and outlet vane angles.

M.H. Shojaeefard, M. tahani, M.B. Ehghaghi, and M.A. Fallahain [5] carried out numerical study of the effect of change blade angle and passage width of centrifugal pump which pumps viscous fluid. In this study 6 different models are modelled with the different outlet angle and passage width of the impeller. The obtained numerical results are compared with experimental and outcome show acceptable agreement between this two. Numerical result show that the impeller with the angle of 30̊ and passage width of 21 mm produce higher head and higher performance compared to other five blade setting. With this arrangement pump efficiency decrease at part load but increase at the best efficiency point and over load performance, compared with other blade design. And its pressure in the outlet area of impeller is higher than other.

E.C. Bacharoudis, A.E Filios, M.D. Mentzos, D.P. Margaris [6] presented the influence of the outlet blade angle on the performance with the help of CFD simulation. He studied that as the outlet blade angle increases the performance curve becomes smoother and flatter. In this study the performance of impeller with same outlet diameter having different outlet blade angles is evaluated and find that, when pumps operate at nominal capacity the head is more than 6%, when outlet blade angle increases from 20° to 50°.

G. Kergourlay, M. Younsi, F. Bakir and R. Rey [6] studied centrifugal pump whose impeller is designed with and without splitter blades. The sliding mesh method use to model the rotor zone motion in order to simulate the impeller volute casing interaction. The flow analysis behaviour shows that the impeller periphery velocity and pressure distribution is more homogeneous when adding slitter blade. Adding splitter has negative and positive effect, it increase head rise and also increase hydrodynamic losses. It decreases pressure fluctuation which relatively decrease pump vibration. The result of global, local experimental and predicted result was found for range of flow rate.

Yang sun sheng, Kong Fan-Yu, Fu-Jian-Hui, Xue Ling [8] carried out CFD study of pump as turbine and did numerical analysis of the model with and without splitter blade. To understand the effects of splitter blades to the steady and unsteady influence of PAT, numerical research was performed. 3D Navier-Stokes solver CFX was used in the performance prediction and analysis of PAT's performance. Results show that splitter blades have a positive impact on PAT's performance. With the increase of splitter blades, its required pressure head is dropped and its efficiency is increased. Unsteady pressure field analysis and comparison show that the unsteady pressure field within PAT is improved when splitter blades are added to impeller flow passage.

Comparison between experimental and numerical results shows that CFD results are in good agreement with those of experimental and can be used in the performance prediction and optimization of PAT. CFD could be used in the optimization and analysis of pump as turbine. Numerical predicted performance comparison of impeller with and without splitter blades shows that PAT's efficiency is increased and its required pressure head is decreased when n splitter blades are added to impeller flow passage. Investigation into unsteady pressure field within volute shows that the absolute pressure gradually decreases along the volute flow channel. The maximum amplitude value of pressure pulsation within volute is located before the volute tongue. When splitter blades are added to the impeller, there is a significant decrease of pressure fluctuation.

Zhang Jinfenge, Yuan shouqi, Fu Yuedeng, and Yuan Jianping [7] carried out a study of CFD simulation of splitter blade on the total flow Field of a low specific Centrifugal pump. Adding splitter blade to the impeller, its periphery velocity and pressure become more homogeneous, which reduce the pressure fluctuation and increase efficiency of pump. The influence of splitter blades are shown that a smoother pressure and velocity distribution at impeller exit and volute inlet, and smaller pressure fluctuation can benefit from the slitter blades. Through the performance analysis, their performance curves show the positive effect of the splitter blades: the operating range is extended, maximum efficiency move to the large flow rate, but not at BEP. The performance curve show that the curve of H-Q become more flat, the operating range is extended, the curve of η-Q become widen and slope of the P-Q curve increase rapidly, which may result in electric overload.

T Shigemitsu, J Fukutomi, K Kaji and T Wada[2], In this research mini centrifugal pump having diameter smaller than 100mm are used. In this main impeller is called TypeC impeller and three other models with the different length splitter blade. Impeller with long, medium and short splitter blade is called TypeG1, TypeG2, and TypeG3 respectively. As result from this they founded that, the head and efficiency of TypeG1 were larger than that of TypeC in the all flow rates.

Sujoy Chakraborty, K.M Pandye and Bidesh Roy[10] investigate the numerical investigation of a centrifugal pump with the different blade number. Centrifugal pump with impeller blades 4,5,6,7,8,910,11 and 12 been modeled and its efficiency at 2700rpm, 3300rpm and 3700rpm is evaluated by CFD code on commercial software Fluent 6.3. The head of centrifugal pump grows all the time with the increase of blade number but the change regulation of efficiency is little complex. But there are optimum values of blade number for each one. So the optimum blade number of the centrifugal pump in this paper for a maximum efficiency is 10.

LIU Houlin, WANG Yong, YUAN Shouqi, TAN Minggao, and WANG Kai [9], In this research the model pump has design specific speed of 92.7 and an impeller with 5 blades. The baled number is varied to 4, 6 and 7 with the casing and other geometric parameter kept constant. Increase of blade number is helpful to reduce the mixture loss, The maximum discrepancy of prediction result of head, efficiency and required net positive head are 4.83%, 3.9%

and 0.36m respectively. Best performance of the pump with 5 blades.

Baoling, ZHU Zuchao and ZHANG Jianci [4] investigate three dimensional turbulent flow in four low specific centrifugal impeller is simulated numerically and analyzed. It is found that the back flow is become smaller in the complex impeller and also pressure rise is uniform from inlet to outlet. From this study experimental and simulation result show that back flow in the impeller is less and it has important influence in pump performance and it effectively solve small flow instability for the low specific speed mid and short blade impeller. The pressure rise uniform inlet to outlet of the impeller and the counters are almost circular and parallel in mid and short blade impeller. The static pressure rise of the 24-blade impeller is the largest in the whole capacity range.

III. CONCLUSION

Different authors have conducted the experimental test and the CFD analysis to improve the performance of centrifugal pump. Impeller geometry mostly effect on the performance of the pump. From the literature survey, following conclusion are made.

- Change in geometry of impeller like, blade angle, blade width, blade exit shape affects the efficiency of the centrifugal pump.
- Adding a splitter blade in the impeller increase the efficiency and head of the pump. And pressure distribution in a pump with splitter blade is more homogeneous.
- Splitter blades are also effective in a pump running in a reverse mode[8].
- With increase in blade number it increase head and flow rate increase continuously. Optimum blade number depends on the pump geometry.

This study can be helpful to predict the performance of a pump, to design the centrifugal pump, with an improvement in efficiency.

REFERENCE

[1] Manivannan, (2010) "Computational Fluid Dynamics Analysis of a Mixed Flow Pump Impeller", International Journal of Engineering, Science and Technology, Vol. 2, No.6, 2010, pp. 200-206.

[2] G. Kergourlay, M. Younsi, F. Bakir, and R. Rey, "Influence of Splitter Blades on the Flow Field of Centrifugal Pump", Hindawi Publishing Corporation International Journal of Rotating Machinery Volume 2007, Article ID 85024, page 13

[3] Yang Sun-Sheng, Kong Fan-Yu, Fu Jian-Hui, and Xue Ling, "Numerical Research on Effect of Splitter Blade to the Influence of Pump as Turbine", Hindawi Publishing Corporation International Journal of Rotating Machinery Volume 2012, Article ID 123093, 9 pages, 2013

[4] Sujoy Chakraborty , K.M Pandye and Bidesh Roy, "Numerical Analysis on Effects of Blade Number Variations on Performance of Centrifugal Pumps with Various Rotational Speeds" ,International Journal of Current Engineering and Technology ISSN 2277 – 4106

[5] E.C. Bacharoudis, A.E. Filios, M.D. Mentzos, D.P. Margaris. "Parametric study of a centrifugal pump impeller by varying the outlet blade angle". The Open Mechanical Engineering Journal; 2, pp.75-83; 2008

[6] M.H. Shojaeefard, M. tahani, M.B. Ehghaghi and M.A. Fallahain, "Numerical Study of the Effect of Change Blade Angle and Passage Width of Centrifugal Pump" Computer & Fluids,2012, pp. 61-70

[7] Zhang Jinfeng Yuan Shouqi Fu Yuedeng and Yuan Jianping, "Influence of Splitter Blade on the Total Flow Field of a Low-Specific Centrifugal Pump", Technology and Research Center of Fluid Machinery Engineering

[8] T Shigemitsu, J Fukutomi, K Kaji and T Wada, "Performance and Internal Flow Condition of Mini Centrifugal Pump With Splitter Blades", Earth and Environmental Science15, 2012

[9] LIU Houlin, WANG Yong, YUAN Shouqi, TAN Minggao, and WANG , "Effects of Blade Number on Characteristics of Centrifugal Pumps", Chinese Journal of Mechanical Engineering Vol. 23,a2010

[10] Boiling, ZHU Zuchao and ZHANG Jianci," The Flow Simulation and Experimental Study of Complex Centrifugal Impeller", Chinese J.Chem. Eng 14(4) 435-441 (2006)

[11] "The Centrifugal Pump" by Grundfos, Research & Technology

[12] Dr. R. K. Bansal, "Fluid Mechanics and Hydraulic Mechanics", 9th Edn, Laxmi Publications LTD, pp 945-960.

