2nd International Conference on Current Research Trends in Engineering and Technology
© 2018 IJSRSET | Volume 4 | Issue 5 | Print ISSN: 2395-1990 | Online ISSN: 2394-4099
Themed Section: Engineering and Technology

A Review on Improvement of Efficiency of Centrifugal Pump by Altering Design Parameters

Parmar Karan[1], Vasava Himanshu[2], Parmar Amit[3]

[1-3]U.G. Student, Mechanical Engineering, Sigma Institute of Engineering, Bakrol, Gujarat, India

ABSTRACT

Nowadays, all kind of Dynamics machineries are being researched for more efficient work and optimum working. Centrifugal pumps are one of them. Centrifugal pumps are used for various applications like drainage and drinking water system, chemical Industries- Catalyst transfer, acid transfer and neutralizing, waste water/Chemicals- Industrial effluents, purifying water, cooing purpose in process industries- paper pulp, chemicals, and pharmaceuticals and in refrigerating systems etc. Centrifugal pumps can be single-stage or may be multistage pumps depends upon the requirement and number of impellers used in the pump. In case of Single stage pump consist one impeller while in multistage pumps the impellers are mounted in the series in . These pumps are very popular because of recent development of high speed electric motors, steam turbines etc. Hence, for more optimal performance the wide range of parameters are there to be work. This article deals with the improvement of efficiency of centrifugal pump through modification in Design parameters. The paper discusses the available material of performance improvement through various parameters and mainly focuses on the researches related to impeller as well as casing.

Keywords: Centrifugal Pump, C.F. Pump, CFD analysis, internal flow analysis, Impeller Design

I. INTRODUCTION

A Centrifugal pump is mechanically operated machinery which is used to transfer the water to certain height under pressure. It is simple in design. A pump is a device that moves fluids (liquids or gases), or sometimes slurries, by mechanical action. Centrifugal pumps operation is by some mechanism and consumes energy to perform mechanical work by moving fluid. Centrifugal pumps are used to transport fluids by the conversion of rotational kinetic energy to the hydrodynamic energy of the fluid flow. The rotational energy generally comes from an engine or electric motor. The two main parts of the pump are the impeller and the diffuser. Impeller, which is the only moving part, is attached to a shaft and driven by a motor. The fluid enters the pump impeller along or near to the rotating axis and is accelerated by the impeller, flowing radially outward into a diffuser or volute chamber (casing), from where it exits as centrifugal action.

As water leaves the eye of the impeller a low-pressure area is created, causing more water to flow into the eye. Atmospheric pressure and centrifugal force cause this to happen. Velocity is developed as the water flows through the impeller spinning at high speed. The water velocity is collected by the diffuser and converted to pressure by specially designed passageways that direct the flow to the discharge of the pump, or to the next impeller should the pump have a multi-stage configuration.

The pressure (head) that a pump will develop is in direct relationship to the impeller diameter, the number of impellers, the size of impeller eye, and shaft speed. Capacity is determined by the exit width of the impeller. The head and capacity are the main factors, which affect the horsepower size of the motor to be used. The more the quantity of water to be pumped, the more energy is required.

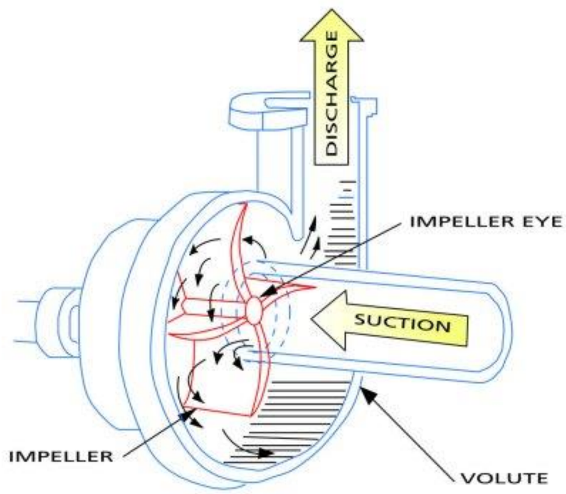

Figure 1 – Centrifugal Pump

These Centrifugal pumps can be analyzed by software code like CFD (Computational Fluid Dynamics).This CFD tool or code helps for optimizing performance of pump. These pumps are working under complex internal flows, which are predicted with the CFD code. The optimized pumps are used for various applications like drainage and drinking water system, chemical Industries- Catalyst transfer, acid transfer and neutralizing, waste water/Chemicals- Industrial effluents, purifying water, in process industries- paper pulp, chemicals etc.

II. COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics is useful for predicting pump performance at various rotational speeds. With the help of numerical simulation mechanical behavior can be analyzed. The prediction of behavior in a given physical situation consists of the values of the relevant variables governing the processes.CFD provides a cost-effective and accurate alternative to scale model frames, stationary and rotating reference frames. The contour and vector plot of pressure and velocity distributions in flow passage are displayed. The simulation results for flow rate and head are compared with analytical formulae. It is concluded that flow pattern of centrifugal pump can be described well with moving reference frame and along with the turbulence model.

CFD applies numerical methods (called discretization) to develop approximations of the governing equations of fluid mechanics in the fluid region of interest.

III. Improvement of Impeller Design a Centrifugal Pump using CFD

Centrifugal pumps are widely used for water supply plants, steam power plants, sewage, oil refineries, chemical plants, hydraulic power service, food processing factories and mines, because of their suitability in practically any service. Therefore it is necessary to find out the design parameter, working conditions and maximum efficiency with lowest power consumption. Study indicates that Computational fluid dynamics (CFD) analysis is being increasingly applied in the design of centrifugal pumps. Various parameters affect the pump performance and energy consumption. The impeller material, blade angle and the blade number are the most critical. Therefore it is necessary for development in the impeller so we can improve the performance of pump. The One-dimensional approach along with empirical equations is adopted for the design of each impeller. The predicted performance curves result through the calculation of the internal flow field. Head-discharge curve play important role into different outlet angles.

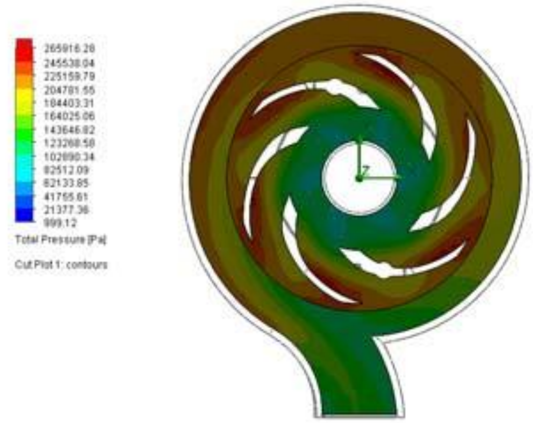

Figure 2 – Design & analysis of pump impeller

The influence of the outlet blade angle on the performance is verified with the CFD. The performance curve becomes smoother and flatter with the increase with the increase outlet blade angle.

At nominal capacity, when the outlet blade angle was increased from 20° to 50°, the head was increased by more than 6% but the hydraulic efficiency was reduced by 4.5%. However, at high flow rates, the increase of the outlet blade angle caused a significant improvement of the hydraulic efficiency. Thus by number of approaches and with trial and error method multiple result may be obtained.

IV. LITERATURE SURVEY

In centrifugal pump, the main design parameters to be concentrated of volute part include base diameter, volute width, vane setting angle of the volute tongue, area of throat. This is manipulated accordance of design in order to achieve better performance. As based on the 2D model the 3D model is obtained by CAD software further this model is carried out in operated in CFD Codes for simulation purposes.

The 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-section. Which are difficult to be predicted for such variant type of flow with the varying parameter in such time. (Honggeng Zhu al, 2012)

The key components like impeller or casing of the centrifugal pump can be redesigned by using the inverse design with singularity method. A cubic Bezier curve was established to express mathematically density function of bound vortex intensity along the blade camber line so as to get the smooth as well as loading carefully controlled blade. The pressure difference and loading coefficient across blade in the original impeller is higher compared to the redesigned impeller. So, by using this approach CFD results confirmed that the impeller hydraulic efficiency was improved by 5% (Wen-Guang LI-2011).

The Validation of present predictions is carried out by comparing with experimental data and with published numerical results. The results of velocity and concentration distribution are verified against rigorous mesh independence. Results in the case pump casing are validated with FEM based numerical results from which the conformance of optimal results are attested. (Krishnan V. -2011)

The main objective design of centrifugal pump becomes to increase the efficiency and at the same time decrease of Net Positive Suction Head (NPSH). To work out for such case centrifugal pumps are numerically investigated using Numeca software. Genetically optimized Group Method of Data Handling (GMDH) type neural networks are used to obtain polynomial models for the effects of geometrical parameters of the pump on both efficiency and NPSH. (H.Safikhani-2011).Such an approach of meta-modeling of those CFD result allows for iterative optimization techniques to design optimally. Thus, simple polynomial models used in a Pareto based multi-objective optimization approach to find the best possible combinations of efficiency and Net Positive Suction Head (NPSH).

Sometimes, research is concerned with rise of head which is affected by change in the outlet blade angle. The systematic research on influence of the various design aspects of centrifugal pump in its performance in the whole range of the flow rates requires numerical predictions and experiment.

The shrouded impellers with outlet blade angle 20 deg, 30 deg as 40 deg and 50deg respectively were designed and performance is predicted. As the outlet blade angle increases the performance curve becomes smoother and flatter for the whole range of flow rates. When pump operates at nominal capacity, the gain in the head is more than 6% but decrease of hydraulic efficiency by 5 %. Moreover, at high flow rates, the increase of outlet blade angle causes a significant improvement of hydraulic efficiency. The minimum radial forces or pressure for that case were calculated near the best efficiency point (BEP) as expected. The same trend is observed for the other two impellers with outlet blade angle β2=30 and 50 deg. It is remarkable the shift of the minimum radial force to higher flow rates (E.C.Bacharoudis-2008).

The flow analysis of 2 or more pumps can be carried out comparatively with the CFD codes like FLUENT which solve the equations or calculations for analysis of turbo machinery flows. These methods are multiple references Frame (MRF), Mixing Plane (MP) and Sliding Mesh(SM) method etc. The steady flow equations can be solved in MRF and MP methods while, unsteady flow equations can be solved in SM method. But, comparatively The Sliding Mesh Method gives good results rather than the MRF and MP methods. Because, these two methods gives the physical approximations and results are found to be far away from the best efficiency point (Erik dick-2001).

With the aid of computational fluid dynamics, the complex internal flow in horizontal split case pump can be well predicted thus facilitating the design of pump. The diameter of impeller, as well as modification in Volute design with the help of CFD design gives better efficiency (V.S.Kadam-2011).

The influence of the outlet blade angle on the performance is verified with the CFD simulation. As the outlet blade angle increases the performance curve becomes smoother and flatter for the whole range of the flow rates. When pump operates at nominal capacity, the gain in the head is more than 5% to 6% when the outlet blade angle increases from 20 deg to 50 deg. However, the above increment of the head is recompensed with 4% decrease of the hydraulic efficiency. Moreover, at high flow rates, the increase of the outlet blade angle causes a significant improvement of the hydraulic efficiency.

V. SCOPE OF THE WORK

The proposed work is designing of impeller, testing of centrifugal pump which will result into the improved efficiency.

VI. CONCLUSIONS

The Hydraulic design of the impeller can be optimized by means of trial and error methods or by means changing the input design of impeller as like Vane angles tangency length as well. From the CFD results mechanical behavior of impeller parts with various parameters like velocity contours .This Contours can be predicted and the optimum design will be manufactured and is to be used for in various applications.

VII. REFERENCES

[1]. DrR.K.Bansal, "A textbook on Fluid Mechanics and Hydraulic machines"

[2]. Sp.asoka, k. Sankaranarayanasamy, t. Sundararajang. Vaidyanathan, k. Udhaya kumar, "pressure drop and cavitations investigations on static helical-grooved square, Triangular and curved cavity liquid labyrinth seals, nuclear engineering and design 241" (2011) 843–853

[3]. VS.Kadam, S.S.Gawade, H.H.Mohite, N.K.Chapkhane, "Design and Development of Split Case Pump Using Computational Fluid Dynamics", Institute of Technology, Nirma University, Ahmedabad.

[4]. Erik dick, Jan vierendeels, Sven serbruyns and John VandeVoorde, "Performance Prediction of Centrifugal Pumps with CFD-Tools", Task Quarterly 5, Volume-4(2001)

[5]. EC. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris, "Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle", the Open Mechanical Engineering Journal, 2008

[6]. Krishnan V. Pagalthivarthi, Pankaj K. Gupta, Vipin Tyagi, M. R. Ravi (2011), "CFD Predictions of Dense Slurry Flow in Centrifugal Pump Casings".

[7]. Wen-Guang LI (April 2011), "Inverse Design of Impeller Blade of Centrifugal Pump with a Singularity Method", Jordan Journal of Mechanical and Industrial Engineering, Volume 5, Number 2,

[8]. Honggeng zhu (2012), rentian zhang, guoqiang luo, bin zhang, "investigation of hydraulic characteristics of a volute-type discharge passage based on CFD",

[9]. Golcu, Mustafa, and Yasar Pancar, 2005, "Investigation of performance characteristics in a pump impeller with low blade discharge angle." World Pumps 2005.

[10]. Aly, S. and S. Al-Zubaidy (1992). "Performance prediction method for centrifugal pumps."

[11]. "Revisited Designing of Intermediate Stage Guide Vane of Centrifugal Pump" XIIIth International Scientific and Engineering Conference "HERVICON-2011"