

ABSTRACT

Nowadays, the centrifugal pumps became very popular because of recent development of high speed electric motors, steam turbines etc. Centrifugal pumps can be single-stage or may be multistage pumps. It depends upon the number of impellers used in the pump. Single stage pump consists of only one impeller while in multistage pumps the impellers are mounted in the series in pumps. These Centrifugal pumps can be analyzed by software code like Computational Fluid Dynamics (CFD). This CFD tool or code helps to optimize the pump performance. The complex internal flows are to be 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, and pharmaceuticals etc.

KEYWORDS: Centrifugal pump, impeller, CF

INTRODUCTION

A centrifugal pump consists of set of rotating vanes called as impellers which are enclosed in housing called as casing. Due to rotation of impeller the fluid from inner radius moves towards the outer radius during this, suction is created at the eye of the impeller. Therefore, continuous lifting of fluid from sump to the pump is carried out and kinetic energy is converted into pressure energy and head is developed from the fluid coming out from delivery pipe.

The increased popularity of centrifugal pumps is due to largely to the comparatively recent development of high speed electric motors, steam turbines and internal combustion engines. The centrifugal pump is a relatively high speed machine and the development of high speed drivers has made possible the development of compact, efficient pumps. Research and development has resulted in both improved performance and new materials of construction that have greatly expanded its field of applicability. It is not uncommon today to find efficiencies of 93%+ for large pumps and better than 50% for small fractional horsepower units.

The mechanical energy is provided with the help of electrically operated motor which is coupled with the pumps. The fluid or water is operated by motor and is forced by the pump to desired work which results into hydraulic energy [1].

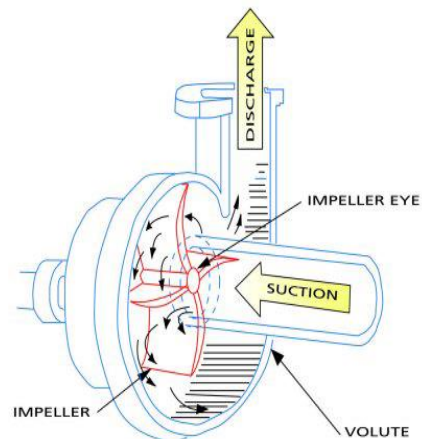


Fig.1. working principle of centrifugal pump

As we are aware of centrifugal pump that these are hydrodynamic machines in which rotating impeller continuously transmits mechanical work from driving machine to fluid. The kinetic energy is converted into potential pressure energy. So it is necessary that to be concentrated on the design and development of impellers as well as volute casing used in pumps.

COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics is very 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

testing with variations on the simulation being performed quickly offering obvious advantages.

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

LITERATURE SURVEY

Nowadays, centrifugal pumps have many applications in industrial fields like chemical industry, petroleum industry, pharmaceutical industry as well as space technology. The centrifugal pump is to be developed gradually for this particular condition of ultra-low specific-speed. In this speed of the pump is much higher and flow rate is much lower. They have studied for specific speed less than 30. The hydraulic model of GSB20-380 type centrifugal pump have designed and developed with ultralow specific-speed (Jie Jin-2012). 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.

Based on hydraulic design model they took 2D model into 3D Solid model along with 5 blades impeller. This solid model from Pro/E is imported to ANSYS and model is meshed and imported to FLUENT and the numerical simulation is carried out as three dimensional incompressible unsteady Reynolds equations is applied. The boundary conditions for area and speed calculation are uniform. The solid wall boundary condition is thermal insulation and impeller is rotating boundary. Thus by using CFD simulation hydraulic model is developed with head and efficiency higher than similar product performance.

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 performance of the centrifugal pump to be tested against the complex internal flow was predicted with the help of Computational Fluid Dynamics (CFD). The standard $k-\epsilon$ turbulence model and SIMPLEC algorithm were chosen for performance prediction. Fundamental equation of fluid dynamics was organized in two reference

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 (Abdulkadir Aman-2011). It is concluded that flow pattern of centrifugal pump can be described well with moving reference frame and turbulence model.

Centrifugal pumps are having wide applications in mining industry, chemical industry, metallurgical operations and coal industry and so on. In typical slurries, solid particles have a range of diameters and concentration depending on the type of slurry for a particular application. Dense slurry flow inside centrifugal pump casing using the Eulerian multiphase model in FLUENT 6.1 is presented. Mixture $k-\epsilon$ turbulence model is used for modeling turbulence. 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 (Krishnan V. - 2011).

As, the centrifugal pumps are group of turbo machines which are used in the large scales in the industries. 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 and 50 deg 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 $\beta_2=30$ 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 6% when the outlet blade angle increases from 20 deg to 50 deg. However, the above increment of the head is recompensed with 4,5% 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.

SCOPE OF THE WORK

The proposed work is to carry out design of impeller, its manufacturing and testing of centrifugal pump which will result into the improved efficiency.

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. From the CFD results mechanical behavior of impeller parts with various parameters like velocity

contours can be predicted and the optimum design will be manufactured and is to be used.

REFERENCES

- [1] Dr.R.K.Bansal, "A textbook on Fluid Mechanics and Hydraulic machines" ,vol-9,ISBN :978-81-318-0815-3,2015
- [2] JieJin^aYingFan^aWeiHan^aJiaxin Hu^b, "Design and Analysis on Hydraulic Model of the Ultra-low Specific-Speed Centrifugal Pump" ^aSchool of Energy and Power Engineering, Lanzhou University of Technology, Lanzhou 730050, China; ^bChina Frist heavy Industries, Qiqihaer 161042, China, International Conference on Advances in Computational Modeling and Simulation, Procedia Engineering Volume 31, (2012) 110 – 114.
- [3] Wen-Guang LI, "Inverse Design of Impeller Blade of Centrifugal Pump with a Singularity Method", Jordan Journal of Mechanical and Industrial Engineering ,Volume 5, Number 2, April 2011, ISSN 1995-6665, Pages 119 – 128.
- [4] AbdulkadirAman, SileshiKore and Edessa Dribssa, "Flow Simulation And Performance Prediction Of Centrifugal Pumps Using CFD-tool" ,Journal of EEA, Volume 28, 2011, Pages 59-65.
- [5] Krishnan V. Pagalthivarthi, Pankaj K. Gupta, VipinTyagi, M. R. Ravi, "CFD Predictions of Dense Slurry Flow in Centrifugal Pump Casings" , International Journal of Aerospace and Mechanical Engineering 5:4 2011, Pages 254-266.
- [6] H. Safikhani, A. Khalkhali and M. Farajpoor, "Pareto Based Multi-Objective Optimization of Centrifugal Pumps using CFD, Neural Networks and Genetic Algorithms", Engineering Applications of Computational Fluid Mechanics Volume-5, No.1,Pages 37-48 (2011).
- [7] E.C. 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, Volume-2, Pages 75-83.
- [8] Erik dick, Jan vierendeels, Sven serbruyns and John VandeVoorde, "Performance Prediction of Centrifugal Pumps with CFD-Tools", Task Quarterly 5, Volume-4(2001), Pages 579-594.
- [9] V.S.Kadam,S.S.Gawade,H.H.Mohite,N.K.Chapkane, "Design and Development of Split Case Pump Using Computational Fluid Dynamics", Institute of Technology, Nirma University,Ahmedabad,08-10 December,2011,Pages1-4.
- [10] Weidong Zhou, Zhimei Zhao, T. S. Lee, and S. H. Winoto, "Investigation of Flow Through Centrifugal Pump Impellers Using Computational Fluid Dynamics", International

Journal of Rotating Machinery, Volume-9(1):
Pages 49–61, 2003.

AUTHOR BIBLIOGRAPHY

