Modelling and Optimization of Centrifugal Pump

Kartik Mandavia[1] Prof. V. H. Chaudhari[2] Prof. M. J. Zinzuvadia[3]
[1]M.E Student [2,3]Associate Professor
[1,2,3]Department of Mechanical Engineering
[1,2,3]B.V.M. Engineering College, V.V.Nagar, Anand, Gujarat, India

Abstract— Centrifugal pumps are widely used in various industrial, agriculture and residential sector where requirements of head and discharge are moderate. They are also a major source of energy wastage. It was estimated that more than 10-15 % of world energy and about consumes 60 % of energy processing plant consume by pumps. Therefore, optimization of pump for improving its efficiency was aim of this research. This research paper mainly focused on optimization of centrifugal pumps impeller and volute casing using Ansys and design of experiments as tool for analysis. CFD analysis will provide effective results for studying different parameters of pump and also for multi parameter optimization of centrifugal pump geometry for getting higher efficiency .The single stage end suction centrifugal pump was consider for analysis. Such pumps were mainly used in processing industries for different type of fluids.

Key words: Single stage centrifugal pump, backward type enclosed impeller, K-ε turbulence mode, Ansys CFX 14.0 software

I. INTRODUCTION

Centrifugal pumps are turbo machines used for transporting liquid by raising a specified volume flow to a specified pressure level. Among all kinds of them, single suction centrifugal pumps are commonly used in industry for several applications due higher efficiency. The majority of market and medium flowing through centrifugal pumps are water or oil. Therefore, growth of one percent of efficiency is of utmost worth. Hence, many researchers are concentrating on improving pump efficiency by enhancing shapes of impeller blades [14]. This study reveals only modification with design parameters of centrifugal pump as a multi objective optimization problem that has been considered so far in this research paper. The optimization of single suction centrifugal pumps is truly a multi-objective optimization problem using multiple design requirements. Most current design and optimization methods for centrifugal pumps implement trial-and-error methods to create and test physical prototypes, which are generally high cost and time-consuming. Multi-objective optimizations were carried out using numerical method and collection of individual objective functions instantaneously [1, 14].

II. LITERATURE REVIEW

K. Pandya C. M. Patel [1] had carried out a study for CFD analysis of centrifugal pump impeller and provided information about different literature paper regarding its optimization with help of CFD. As per this paper by Jekim. j. Damor et al. [15]., they conducted experimental investigation on centrifugal water pump with an impeller of outlet 111 mm diameter, backward curved blades, formal discharge of 4.00 lps and 12 m of head to assess consequence of various operating states like head, discharge, power and speed on performance of a pump. Further an impeller was modelled using solid works software and computational fluid dynamics (CFD) analysis was carried out using Ansys CFX software on developed model of an impeller to predict performance, virtually and to verify with experimental results of pump. It was found that CFD analysis will be better tool for analysis than experiment.

A.P. Singh, Sujoy Chakraborty [2] had performed CFD analysis for pump having inlet blade angle 25º and outlet blade angle 33º. They investigated head and discharge at 2500 rpm using 7, 8, and 9 numbers of blades and checked, which were best optimum for that pump. They had used standard k-ε turbulence model by using Ansys fluent 6.3 software. They also performed the grid refining test for find optimum meshing size which brings lesser variation in results. It was found that with increasing of blade number, reduces mixture loss of 'jet' & 'wake' in centrifugal pump and also there was growth of area of low pressure region at suction of blade [2, 4]. The static pressure was gradually increasing and total pressure too. Uniformity of static pressure distribution at suction section become worse and worse, while at diffuser section, it became better and better.

Budea sanda, Carbune varzaru Daniela [3] investigated influence of blades angles on radial impeller geometry of a centrifugal pump. Investigation on influence of an inlet and outlet angles of blades over hydraulic channel size, number of blades and over the blades angular extension was carried out. They provided relation between inlet blade angle and other parameters like breadth of impeller at inlet, number of the blades depending of inlet angle of impeller. They have modified inlet and outlet blade angle for this study using Ansys for visualizing streamlined flow [3, 6, and 8].

LIU Houlin, WANG Yong, YUAN Shouqi, TAN Minggao, and WANG Kai [4] had investigated effect of different blade number for pump having flow rate of 50 m^3/hr. and head of 32 m. Fluent 6.1 was used to simulate inner flow field under non-cavitation condition. The standard k-ε turbulence model and SIMPLEC algorithm were applied to solve Reynolds Average Navier stroke equations. From their study they found that with increase blade number the jet and wakes in pump were reduced and also it was observed an effective reduction of cavitation in an impeller of pump [2, 4]. There will increase in head with increase in blade number with reduction in the efficiency.

H L Liu, M M Liu, L Dong, Y Ren And H Du [5] investigated effect of different turbulence model on accuracy of pump anlysis. They carried out analysis for flow rate of 20.31m^3/h, head 46.35m, RPM 2900 rev/min, and prepared modeled in pro–e and carried out CFD anlysis with help of Ansys CFX and founded optimum meshing size which had less deviation from an experimental readings. They also investigated effects turbulence models on CFD numerical simulation of centrifugal pump. Six groups of mesh numbers range from 1 million to 24 million were

meshed.They also carried out the analysis with different turbulence models like RNG, k-epsilon model and k-epsilon model, the absolute discrepancy of efficiency under k epsilon ESARM from their experiments they concluded that k-ε will having lesser deviation form their experimental results. There will be 5 % deviation with different turbulence models.

M.G.Patel, A.V.Doshi [6] investigated the effect of different blade angle on performance of the centrifugal pump using CFD codes. This paper gave brief idea about the GULICH model for various coefficients like different hydraulic losses in volute as well as impeller. They use three different pumps having different configuration and by varying the blade outlet angle they have predicted their effects on the efficiency .They found that blade angle having significant effect on the pump performance [3,6].

Zhigang Zuo, Shuhong Liu,Yizhang Fan and Yulin Wu Hindavia [7] Carried out the fem as well as CFD analysis for ultra-supercritical and supercritical boiler pump which operates at higher temperature .They carried out analysis at NTHP in the fem and found effective thickness for pump casing with respect for different pump volute diameter and material. They also provided the correction factors if the pump was operated at HTHP .For CFD analysis using SST k-ω turbulence model for steady flow calculations and validate the pump results with the experimental one.

Volute diameter was considered for study by varying it and finding optimum one. They carried out fatigue analysis in Ansys workbench with 100,000 grid cells for fem pump was treated as the pressures vessel.

M.H. Shojaeefard a, M. Tahani a, M.B. Ehghaghi b, M.A. Fallahian a, M.Beglari [8] Investigated effect of geometric parameters like blade outlet angle and width with different viscosity fluid. Analysis was done on pump single axial suction and vane less volute casing; impeller of 209 mm in outside diameter and six backwards curved blades. SST k-ε turbulence model was used for the numerical investigation of flow inside the centrifugal pump. They also did experiment for validation of their results. They found that the friction on impeller in the case of oil decreases the head and efficiency and increases the power consumption compared with the case of water. On the other hand, the performance of centrifugal pumps drops sharply during the pumping of viscous fluids i.e. oil. By changing the original geometry of the impeller will improves the centrifugal pump performance. There will be reduction in dissipation arising by vortex formation with increase in impeller passage when the pump handles viscose liquid. The width of wake at the outlet of impeller and the hydraulic losses decrease at BEP point [13,8].

Mona Golbabaei Asl , Rouhollah Torabi , S. Ahmad Nourbakhsh [9] carried out fem analysis on the pump casing have two different material i.e. steel and cast iron having different thickness nonlinear static analyses were performed to determine the maximum failure-equivalent stress in cast iron casing. They found that even the thickness of steel casing was less the crack penetration in case of cast iron will more savior. Maximum axial, radial, tangential stress in case of steel will less compare to CI .They have valid their results by doing experimental results of hydrostatic test. Working life of steel will far better than CI.

Daniel O. Baun, Ronald D. Flack [10] has carried out experimental investigation of comparing performance of three different type of pump casing. For this they taken spiral, concentrated, double volute casing for two impeller for four and five blade .They found that Impeller with four blade having maximum efficiency will with spiral casing then double and concentrated casing. Were as the radial force magnitude higher in double volute then concentrated, then spiral in decreasing order. but in case of five blade impellers hydraulic performance will be in higher in double then concentrated and spiral in decreasing order so this conclude that even though volute are same but their will effective change with blade number but there will no such change in case of radial force magnitude.

The losses in the double volute will increase over the single volute because of more wetted surface and two tongues, which result in twice the incidence losses as compared to the single volute. It will also add blockage in the volute, thus increasing the low velocities at any given flow rate. This has the combined effect of increasing skin friction losses and also shifting the matching point of the impeller-volute combination to a lower flow rate. The concentric volute has flat head characteristics and therefore an improvement in stability over a baseline spiral volute

Ling Zhou National RCP&PSE [11] carried out both experiment and also CFD analysis for investigating effect of rear shrouding of impeller on hydraulic performance and axial force. Using CFD they founded that for head for shroud radius R57.5 will having higher head as compare to R45.5 so pump head performance will also affected due to shrouding .but for case R55.1 will having higher efficiency compare to R57.5 as it will having less resistance to flow but even though the radius of R45.5 was less than R57.5 its efficiency was lower because of gap leakage. For the analysis, Ansys ICEM 13.0 software for meshing with total grid elements number approximately $2.5x10^6$.

For same impeller they had taken three different shrouding diameter (a) R55.75mm (b) R55.4mm (c) R54.55mm for experiments to study axial forces. They founded that the axial force for R57.5mm decreases most sharply when the flow rates rise. Due to the small impeller outlet area, the streams in the diffuser have a high velocity and cannot convert to the pressure energy ideally, and a vortex was found in the middle of diffuser .For R54.1 mm, the static pressure of the flow increases gradually from impeller inlet to diffuser outlet, and the change trend was relatively uniform. In contrast, the smallest rear shroud of R45.5mm has lowest static pressure due to an huge reduction of impeller diameter, which will lead the impeller output Energy decrease greatly.

Sun-Sheng Yang, Fan-Yu Kong, Hao Chen, Xiang-Hui Su [12] improves the efficiency of a pump as turbine (pat as blade wrap angle is one of the key geometric parameters in impeller design. They investigation effect of blade wrap angle on pump performance. They carried out CFD analysis for three different pumps having different specific speed and also carried out experiment for verifying their test results. From their study they founded that with increase in the wrap angle the head losses will effectively reduce. There will be maximum hydraulic loss in impeller volute gap and. For their analysis Ansys-CFX 12.0 was selected for the solution of the 3d Navier-stokes equation due to its

characteristics of robust and fast convergence. The turbulence model selected was the k-ε model. Meshing done in Ansys ICEM-CFD 12.0. They conclude that the flow rate at the best efficiency point (BEP) will increased with the decrease of the blade wrap angle.

M.H.shojaee and F.A Boyaghaci [13] use Ansys Fluent to carry out 2d analysis of pump to investigate effect of viscous fluid on pump. They use k-ε turbulence model for their study and pressure and velocity coupling calculated through SIMPLEC algorithm for water, oil (43x10-6 m^2/s and 62x10-6 m^2/s) .They founded that when handling viscous fluid disk frictional loss will increase whereas jet and wakes[2,4,13] will reduces at blade pressure side.

III. CONCLUSION

CFD is better tool for optimization then experimentally optimization of pump as it is less costly and time consuming [1, 14]. pump can be multi-parameter optimized as more than one parameters are affecting on pump efficiency [1,12,14].Based on Literature Survey main parameters for optimization will be blade number[2,4] ,blade angle [3,6,8], blade width [3,7] , volute diameter ,Wrap angle[12]. Thus experimentally validated CFD model will be done and then this analysis will be used for optimization of pump using Taguchi orthogonal array method. For finding out significant optimum combination of parameters. Analyses will be done using Ansys CFX 14.0 [5, 11, 12 and 15] using k-ε turbulence model [5] with optimum number of meshing size.

IV. ACKNOWLEDGEMENT

Thanking to Guides for carried out the literature survey and also for entire project.

REFERENCES

[1] K. Pandya, C. M. Patel, "A Critical Review On CFD Analysis Of Centrifugal Pump Impeller", IJAERD ,Vol. 1 issue 6, 2014, pp.16-25.
[2] K.M. Pande, A.P. Singh and Sujoy Chakraborty,"Numerical Studies On Effects Of Blade Number Variations On Performance Of Centrifugal Pumps At 2500 Rpm", JEAD, Vol. 6 No. 3A, 2012, pp. 863-868.
[3] Budea sanda , Carbune varzaru Daniela," The influence of the inlet angle over the radial impeller geometry design approach with Ansys" ,JESR, Vol. 18, No. 4,2012, pp.32-39.
[4] LIU Houlin, WANG Yong, YUAN Shouqi, TAN Minggao, And WANG Kai, "Effects Of Blade Number On Characteristics Of Centrifugal Pumps" CHME, Vol. 23, 2010, pp. 1-6.
[5] H L Liu, M M Liu, L Dong, Y Ren And H Du , "Effects Of Computational Grids And Turbulence Models On Numerical Simulation Of Centrifugal Pump With CFD" ,26th IAHR Symposium On Hydraulic Machinery And Systems by IOP Publishing , pp. 1-7.
[6] M.G.Patel, A.V.Doshi, "Effect of Impeller Blade Exit Angle on the Performance of Centrifugal Pump", IJETAE, Volume 3, issue 1, January 2013, pp.702-706.
[7] Zhigang Zuo, Shuhong Liu, Yizhang Fan, and Yulin Wu1 Hindavia , "Optimization of a Centrifugal Boiler Circulating Pump's Casing Based on CFD and FEM Analyses" 2014, pp.1-10.
[8] M.H. Shojaeefard a, M. Tahani a, M.B. Ehghaghi b, M.A. Fallahian a, M. Beglari , "Numerical study of the effects of some geometric characteristics of a centrifugal pump impeller that pumps a viscous fluid" ,ELSEVIER 2012, pp.62-70.
[9] Mona Golbabaei Asl , Rouhollah Torabi , S. Ahmad Nourbakhsh , "Experimental and FEM failure analysis and optimization of a centrifugal-pump volute casing" ELSEVIER 2009, pp.1996-2003.
[10] Daniel O. Baun, Ronald D. Flack, "Effects of Volute Design and Number of Impeller Blades on Lateral Impeller Forces and Hydraulic Performance", International Journal of Rotating Machinery Taylor & Francis Group, 2003, pp. 145-152.
[11] Ling Zhou National RCP&PSE, "Numerical and Experimental Study of Axial Force and Hydraulic Performance in a Deep-Well Centrifugal Pump with Different Impeller Rear Shroud Radius", Jiangsu University, Journal of Fluids Engineering, ASME, 2013, Vol. 135, pp.1-8.
[12] Sun-Sheng Yang, Fan-Yu Kong, Hao Chen, Xiang-Hui Su, "Effects of Blade Wrap Angle Influencing a Pump as Turbine" Journal of Fluids Engineering 2012, ASME, pp. 1-6.
[13] M.H.shojaee and F.A Boyaghaci , "Studies on the influence of various Blade outlet angle on Centrifugal pump when handling Viscous fluids", American Journal of Applied Science 2007,pp.718-724.
[14] Yu Zhang, "Multi-objective optimization of double suction centrifugal pumps using Kriging Meta models" Advances in Engineering Software 74, Elsevier 2014 , pp.16-26.
[15] Jekim J. Damor, Dilip S. Patel, Kamlesh.H.Thakkar, Pragnesh K. Brahmbhatt "Experimental and CFD Analysis of Centrifugal Pump Impeller- A Case Study", IJERT 2010, pp. 1-7.

