IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X,
PP 12-16
www.iosrjournals.org

Review of optimization of mixed flow impeller using Ansys CFX

Prof. Prashant.S. Raut [1], Prof.Deepak.K.Patil [2] , Prof.Gopal A.Wadnere[3]
[1,2,3](Mechanical, Sandip Polytechnic, Nashik)

Abstract: In this paper, the results from numerical studies of the mixed flow through a mixed flow pump is presented here and compared with the measurements. The final results matches well with measurements at best efficiency flow conditions. The entire range of H-Q characteristics is predicted and has matched well with the measurements. The predictions for efficiency at off design flow conditions are seen to be deviating from the one actually measured. Some more investigation needs to be done in this direction. Similar exercise is being carried out for pumps with different specific speeds. CFD analysis with ansys cfx is an effective tool to calculate quickly and inexpensively the effect of design and operating parameters of the pump. with properly designing pump impeller the efficiency of pump can be improved.

Keywords: *mixed flow impeller, Computational fluid dynamics analysis of a mixed flow pump impeller, SolidWorks 2009*

I Introduction

A pump is device which is used to displace fluids or sometimes slurries by mechanical work done. Pump is a mechanical positive device generally used for lifting liquids from a lower level to a higher. This will achieved by creating low pressure at the inlet and high pressure at the outlet of the pump. However, actual work has to be done by the prime movers to enable it to give mechanical energy to the liquid which converts into pressure energy. It is mostly in used in industries and home applications. Centrifugal pumps are the device, which employies centrifugal force to lift from a lower level to a higher level by developing pressure.The centrifugal pump can moves liquid by rotating impellers inside a volute casing. The liquid is passing through the casing inlet to the center of the impeller where it is picked up by the impeller vanes. In the mixed flow pump, the addition of energy to the liquid occurs when the flow of the liquid in an axial as well as in the radial directions. In this type of pump liquid through impeller is a combination of axial and radial directions The head can be developed partly by the action of centrifugal force and partly by the action of propelling force. These pumps mostly suitable for irrigation purposes where large amount of water at a lower head. In this pump, addition of energy to the liquid occurs.

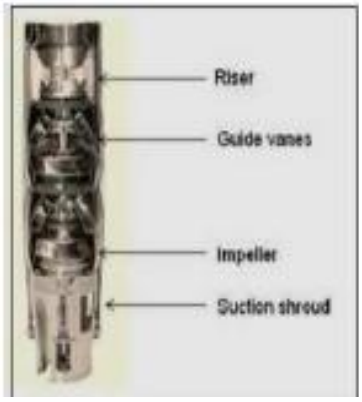


Fig 1: Mixed Flow Pump

I.I Vertical type Mixed flow Pump and Working Principle

The vertical flow type mixed flow pumps are mainly consists of four main subassemblies. These main subassemblies are driver, upper discharge head, body with column assembly and the flow pump assembly. The main pump assembly is also consists of various parts which are as shown in Figure1. The input power is supplied by the electric motor or any other type of driver such as two stroke diesel engine to the pump.

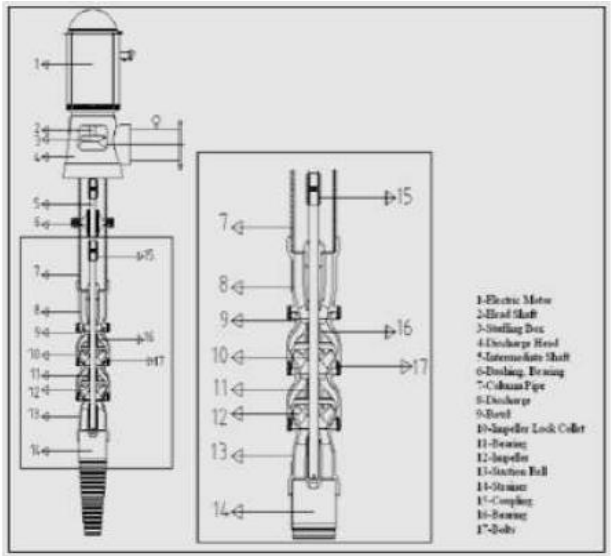


Fig2:- Parts of the mixed flow Pump Assembly

II Review of Past Work

A.Manivannan[1] h**e** studied the Computational fluid dynamic analysis of a mixed flow impeller by comparing existence with new results..

Fig 3:- Model of mixed flow impeller

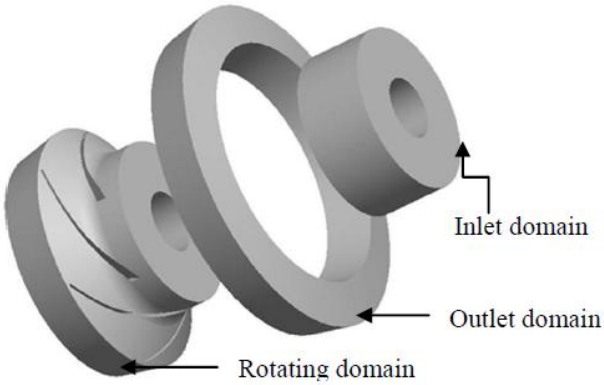


Fig 4:-Water domain Model of mixed flow impeller

The given model of the mixed flow impeller is as shown in Fig.3. The computational domains contains the single passage to the rotor,the inlet domain and outlet domain. The related model is as shown in Fig.4. The final meshing of rotor and stator was generated by using commercial software Star Design3.2. The total number of nodes obtained are 81000 and 32000 for the rotor and the given stator respectively. The current computation is carried out with a multiple frame of reference approach because the impeller flow field is in a rotating frame and the intake section and outlet sections are in stationary frame. The meshes of three computational domains, the intake section, impeller and outlet, are generated separately in the analysis. The detailed experimental study of the hydraulic efficiency of mixed flow pump was calculated by the cfd analysis . New Test results are used to obtain the boundary conditions and it is used to find out the best efficiency for the mixed flow impeller.

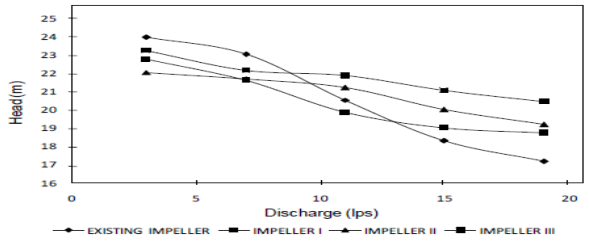


Fig5:- Head develope by the existing and modified impellers

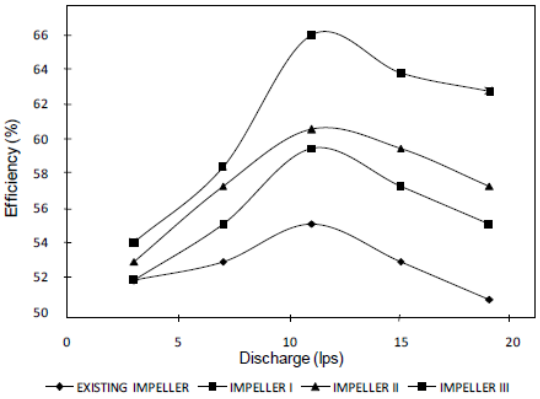


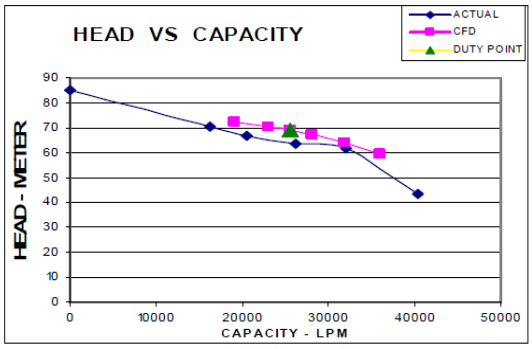
Fig6:- Efficiency of the existing and modified impellers

The detailed design parameter and CFD analysis of the mixed flow impeller of pump, the following conclusions are getting. the mixed flow pump having the best efficiency point of the pump is to be found out to be 11 lps. For the given impeller, the head, power ratings and efficiency of impeller are found out is 19.24 m, 9.46 kW and 55% respectively.For the impeller 1, the percentage of increase in the head, power ratings and efficiency are 3.22%, 3.9% and 7.27% respectively. For the impeller 2, the percentage change in the head, power ratings and efficiency are 10.29%, 7.61% and 10.91% respectively. For the impeller 3, the percentage increase in the head, power rating and efficiency are 13.66%, 12.16% and 18.18% respectively.

Impeller design	Inlet angle (deg)	Outlet angle (deg)
Existing	75	55
Impeller 1	75	60
Impeller 2	65	60
Impeller 3	55	65

From the above table it can be concluded that impeller 3 gives better performance. No other tool is effective and quick than CFD analysis to find the effect of design and operating parameter of pump. The efficiency of pump can be improved by properly designing pump impeller.

Kiran Patel and N.Ramakrishnan[2] Reviewed the CFD analysis of mixed flow pump In this Paper, Author says that,



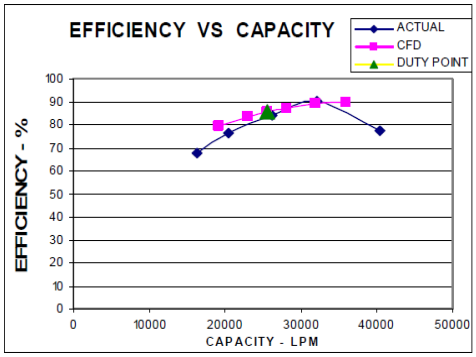


Fig7:-Head vs capacity curve **Fig8:- Efficiency vs capacity curve**

Head obtained by CFD analysis is 5 to 10 % higher than the test result at rated point. the Power expected by CFD analysis is 5 to 10% higher. compared with the test result, disc-friction power loss calculated using the NEL method is added to CFX analysis generated power for calculating the total power requirement.

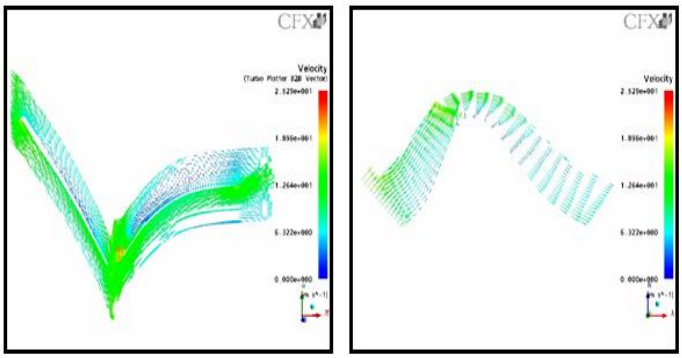


Fig:-[A.] Velocity vector plot in Blade-to-blade view [B] Velocity vector plot in Meridional view

The Efficiency expected by the CFD analysis is little higher than the test result.some losses like Leakage-loss is predicted by using NEL. The mixed flow Pump efficiency by considering disc friction loss and leakage-loss is predicted and it was found within +5% range, at duty point.The nature of Pump operating characteristic curves of the CFD analysis is the same as that of the standard curves.CFD analysis indicates recirculation zone in the stator at duty point, which will caused reduction in efficiency. Efficiency can be improved by 1% after matching stator angle and by changing hub curve profile.Stator blade loading at hub and shroud will be improved.

Mandar TABIB[3], studied the CFD analysis of mixed flow impeller.A CFD code, the ANSYS® CFX® 12.1, was used to get the head and the pressure, velocity streamlines.The analysis results to show the head of 7.45m and the head obtained by the experimental work in industries was 8.08 m. The efficiency obtained by experimental result was 53.27 % and by CFD analysis 49.6 %.By the CFD analysis high values are obtained for the head, compared to the manufacturer experimental head. In the CFD analysis there is no other parameters from the diffuser, so that the friction losses are comparatively smaller and it will affects the pressure fields and increase the head values. That fact is representing the necessity to show the friction losses due to joining between the diffuser and the given impeller.The final result shows pressure in the impeller channels increasing from the entrance to the discharge in the successive ranges.

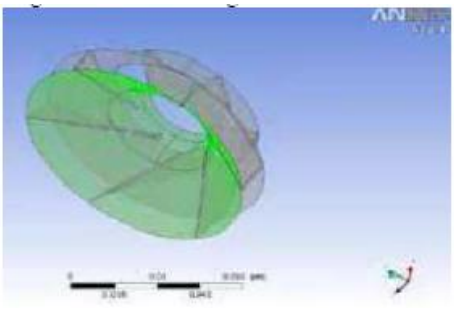


Fig 9:-Boundary condition applied to the pump impeller.

Mitul G. Patel[4] studied the CFD analysis of mixed flow pump and concluded that the mixed flow pump calculation of the flow solutions in a centrifugal pump impeller.

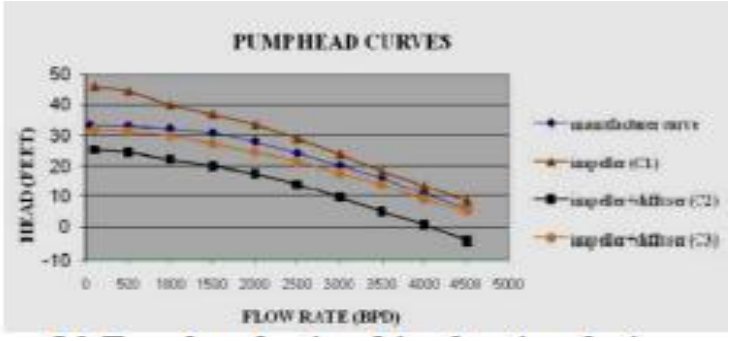


Fig10:-head vs flow rate curve of impeller

In this paper a simulation of the centrifugal pump the internal flow was implemented. The ANSYS® CFX® 11.0, was used to obtain the head performance curve and to calculate the interface connection between the pump parts, the impeller and the diffuser.The boundary conditions will be adjusted in the software to characterize the three dimensional problem. The MODIFICATIONS are done in a model, in order to adjust the geometry to the software limitations,the numerical analysis by using a CFD code, ANSYS® CFX® 11.0 shows results in agreement with the given references.The three-dimensional simulation of the impeller interaction of a hydraulic pump.The results obtained for the pressure fields and therefore the head performance curves were satisfactory in the three conditions which are tested.

III Conclusion

The first paragraph under each heading or subheading should be flush left, and subsequent paragraphs should have a five-space indentation. A colon is inserted before an equation is presented, but there is no punctuation following the equation. All equations are numbered and referred to in the text solely by a number enclosed in a round bracket (i.e., (3) reads as "equation 3"). Ensure that any miscellaneous numbering system you use in your paper cannot be confused with a reference [4] or an equation (3) designation.

References

[1]. Manivannan. "Computational fluid dynamics analysis of a mixed flow pumps impeller". International Journal of Engineering, Science and Technology, vol.2, no.6, pp.200-206, 2010

[2]. Kiran Patel 2006, CFD analysis of mixed flow pump, International Journal of Advanced Engineering Research and Studies.

[3]. Furukawa, H. Takahara, T. Nakagawa, Y.Ono,Pressure Fluctuation in a Vaned Diffuser Down stream from a Centrifugal Pump Impeller, Intl. J. of Rotating Machinery, Vol. 9, 2003, pp. 285-292..

[4]. Mitul G. Patel, Bharat J. Patel. "CFD analysis of mixed flow submersible pump impeller". Indian Journal of applied research, vol.1, issue 9, pp.97–100, June. 2012

[5]. Dr. H.S. Shivashankar. "Design and CFD analysis of single stage, end suction, radial flow centrifugal pump for mine dewatering application". IJREAS, vol.2, issue 2, pp.6-18, Feb. 2012.

[6]. Dr.K. Purushothaman. "Analysis of a centrifugal pump impeller using ANSYS-CFX". International Journal of Engineering Research and Technology, vol.1, issue 3, May. 2012.