Performance, Optimization and CFD Analysis of Submersible Pump Impeller

Ankurkumar. H. Vyas[1]
[1]M.E (CAD/CAM) student
[1]Indus Institute of Technology and Engineering, Rancharda, Ahmedabad, Gujarat, India

ABSTRACT—To improve the efficiency of submersible flow pump, Computational Fluid Dynamics (CFD) analysis is one of the advanced tools used in the pump industry. A detailed CFD analysis was done to predict the flow pattern inside the impeller which is an active pump component. From the results of CFD analysis, the velocity and pressure in the outlet of the impeller is predicted. CFD analyses are done using ANSYS CFX software. In this research paper we will modified the impeller design by choosing some parameter.

I. INTRODUCTION

A wide variety of centrifugal pump types have been constructed and used in many different applications in industry and other technical sectors. However, their design and performance prediction process is still a difficult task, mainly due to the great number of free geometric parameters, the effect of which cannot be directly evaluated. The significant cost and time of the trial-and-error process by constructing and testing physical prototypes reduces the profit margins of the pump manufacturers. For this reason CFD analysis is currently being used in the design and construction stage of various pump types. From the CFD analysis software and advanced post processing tools the complex flow inside the impeller can be analyzed. Moreover design modification can be done easily and thus CFD analysis reduces the product development time and cost. The complex flow pattern inside the centrifugal pump is strong three dimensional with recirculation flows at inlet and exit, flow separation, cavitation. Also the efficiency of the impeller can be improved by changing the volute design of the impeller and by increasing the number of impeller blades.

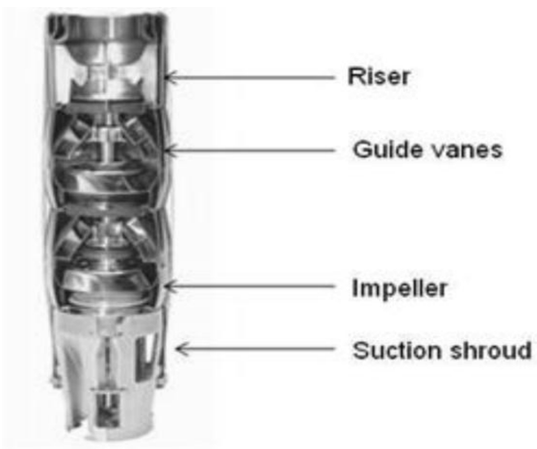

Fig (1): Submersible pump

II. LITERATURE REVIEW

A.MANIVANANA [1] has done existing impeller is modified to get better results. The optimum value of impeller is calculated by using the empirical relations. The rotating impeller imparts energy to the fluid. It is the most important, the only rotating element in the pump. The performance of the pump is greatly depends on the impeller performance. Efficiency of the impeller depends on flow condition inside the impeller and geometric features of the impeller. The flow conditions inside the impeller can be varied by changing the geometric features of the impeller. As the result of impeller modification are as below.

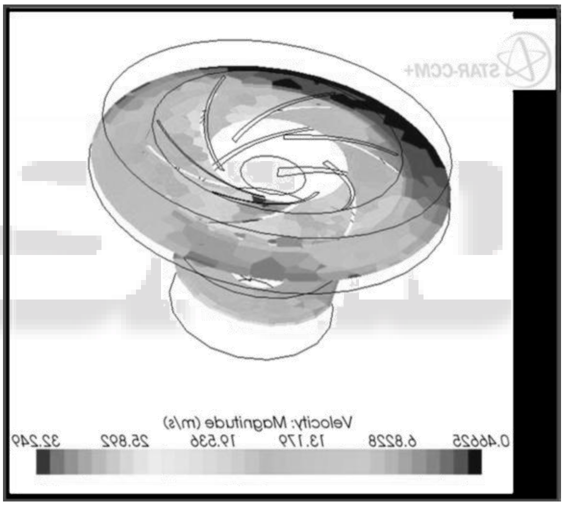

Fig (2)

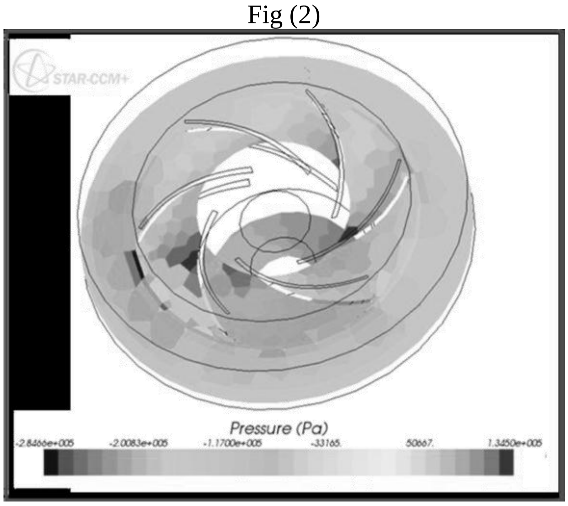

Fig (3)

Fig (2), (3): Velocity contours at 5 m/s inlet velocity

As the flow entering the impeller eye, it is diverted into the blade-to-blade passage. Due to the unsteady effect developed at upstream, the flow entering the passage is no longer tangential to the leading edge of impeller blade. Separation of flow can be observed at all passages leading edge. Increased flow velocity can be observed at the blade inlet due to the blockage of the flow, whereas on the contrary the pressure is reduced. Further downstream the contours become smooth between the blades and the pressure increases continuously towards the exit of the computational domain. The hydraulic test was done on existing impeller and the results are compared with modified impeller CFD results. The k-e turbulence model can be used to analyze the mixed flow impeller and area averaged value of pressure and velocity can be used to calculate the head, efficiency and power rating of the impeller.

The existing impeller, the head, power rating and efficiency are found out to be 19.24 m, 9.46 kW and 55% respectively. the impeller 1, the percentage increase in the head, power rating and efficiency are 3.22%, 3.9% and 7.27% respectively. the impeller 2, the percentage increase in the head, power rating and efficiency are 10.29%, 7.61% and 10.91% respectively. the impeller 3, the percentage increase in the head, power rating and efficiency are 13.66%, 12.16% and 18.18% respectively. Based on the above it is concluded that impeller 3 gives better performance.

Thus CFD analysis is an effective tool to calculate quickly and inexpensively the effect of design and operating parameter of pump. By properly designing pump impeller the efficiency of pump can be improved.

Kiran Patel and N Ramakrishnan [2] study about CFD analysis of mixed flow impeller. The results of steady state analysis of original geometry and comparison of test result and CFD analysis result. The comparison of results of original and modified geometry was presented. This analysis is based on stator and rotor effect.

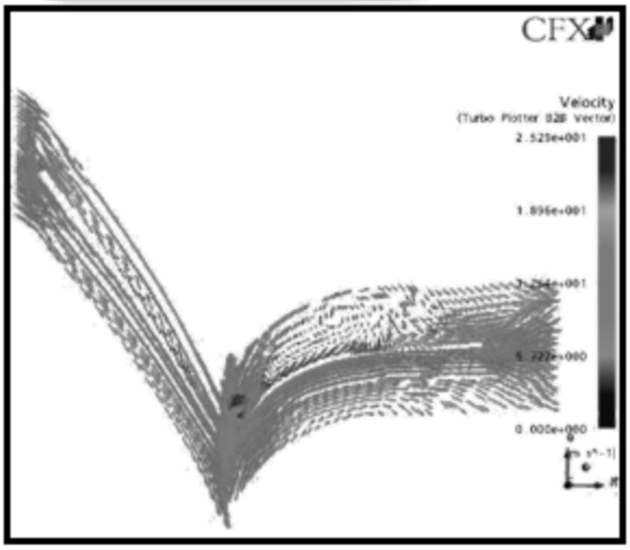

Fig (4)

Here vectors are streamlined in rotor; but in stator due to secondary flow vectors are recirculating. Total pressure in stationary frame is plotted in blade-to-blade view at duty point in. Total pressure is gradually increasing in rotor and loss took place in stator.

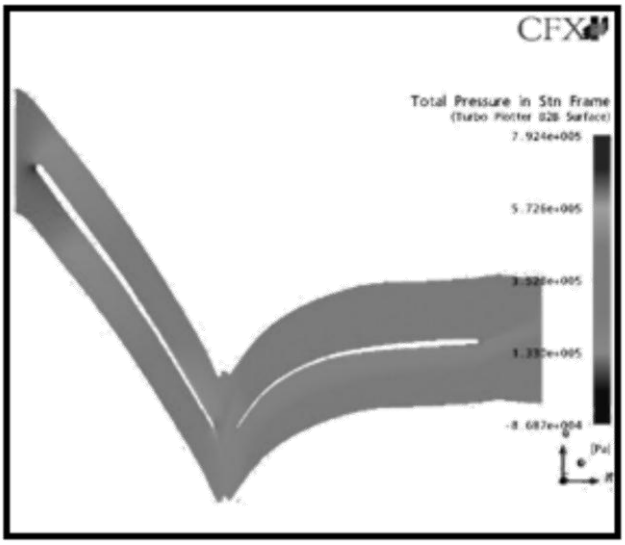

Fig (5)

Head predicted by CFD analysis is 5 to 10 % higher than the test result at rated point. Power predicted by CFD analysis is 5 to 10% higher at rated point. Efficiency predicted by CFD analysis is higher than the test result. Efficiency is improved by 1% after matching stator angle and changing hub curve profile.

Shahin, Ayad, Samir S. [3] study about performance and unsteady flow field prediction of a centrifugal pump with CFD tools.

There analyses results are as under below.

In FLUENT to investigate the unsteady flow in radial pumps with different cases: volute, van less and vanned diffuser. Calculations are performed at different operating points, for the impeller and diffuser.

The unsteady calculations combined with the sliding mesh technique have proved to be a good tool to investigate rotor –stator interaction in centrifugal pump. The pump internal flow field is investigated by using numerical methods and compared well with experimental data over the wide flow range.

For pump with volute casing, uniform pressure increase with radial direction through the different impeller channels is maintained axis-symmetric for the nominal flow rate, whereas for off-design conditions 0.45 Q_{design} a region with higher pressures is found just preceding the volute tongue, in the rotating direction, while there is a region of low pressure distribution is observed just following the volute tongue. The pressure fluctuation on the impeller pressure side is much bigger than on the impeller suction side, and it is more evident at the rear part of the impeller blade.

Perez J. †, Chiva S.†, Segala W*., Morales R.*, Negrao C.* [4] study about performance analysis of flow in a impeller-diffuser centrifugal pumps using cfd :simulation and experimental data comparisons.

In this work was simulated numerically the flow inside the first stage of a centrifugal pump composed by two stages. A single phase turbulent flow was considered in transient regime with water under constants proprieties as fluid. Four angular velocities for the rotor were simulated 1150, 1000 and 806 rpm.

From the numerical simulation curves of pressure gauge were elaborated on the rotor and diffuser depending on the mass flow rate. a good agreement was obtained compared with the experimental results obtained in the pump bench of the Unicam University. The pressures obtained in the numerical simulation were slightly under-predicted. The differences of pressure between experimental result and numerical results in the diffuser were bigger, an explanation for this can be found in the election if the turbulent model. It is known that turbulent model was not made for flow on curve surfaces, and it is not describing properly recirculation and swirling flow, in fact the angular velocity does not appear explicitly in the equation obtained by adding the normal stresses. Thus the model is totally blind to rotation effects. The swirling flow can be regarded as a special rotational effect with the axis usually aligned with the mean flow direction.

Maitelli,c.w.s dep.1 they all are Study died about Artificial lift methods can be implemented in the oil industry in order to increase the production rate flowing oil or gas wells. Electrical Submersible Pumping (ESP) is a complex and viable method of artificial lift that is applicable to a wide range of flow rates. ESP systems are responsible for the highest amount of total fluids produced by any artificial lift method. This paper presents a 3D simulation of the stationary flow in the impeller and stator of a mixed centrifugal pump using Computational Fluid Dynamics (CFD) techniques and a commercial software, ANSYS® CFX® Release 11.0. Three conditions were simulated to obtain the pressure fields in the impeller and stator in a stage of the pump. Comparisons among the simulations of the present work and the head capacity of the performance curve given by pump manufacturer were performed and showed satisfactory agreement

III. RESULT

As we can see in the literature survey different authors have conducted the experiments on the impeller and conducted the fluid analysis. Kiran patel and N ramakrishan had done in experiment Efficiency predicted by CFD analysis is higher than the test result. Head predicted by CFD analysis is 5 to 10 % higher than the test result at rated point. A. manivanan has done that CFD analysis is an effective tool to calculate quickly and inexpensively the effect of design and operating parameter of pump. By properly designing pump impeller the efficiency of pump can be improved. This paper will help to design the impeller which could have the more effective which will increase the efficiency of the submersible pump.

REFERENCES

[1] T.E.Stirling : “Analysis of the design of two pumps using NEL methods”
[2] Centrifugal Pumps-Hydraulic Design-I Mech E Conference Publications 1982-11, C/183/82.
[3] Baun D.O., Flack R.D. 2003. Effects of volute design and number of impeller blades on lateral impeller forces and hydraulic performance, International Journal of Rotating Machinery, Vol. 2, No. 9, pp. 145-152.
[4] Cheah K.W., Lee T. S. 2007, Numerical flow simulation in a centrifugal pump at design and off-design conditions, International Journal of Rotating Machinery, Vol. 2007, Article ID 83641, doi: 10.1115/2007/83641, 8 p.
[5] Dupont P. 2003, Cavitating flow calculations in industry, International Journal of Rotating Machinery. Vol. 9, No. 3, pp. 171-179.
[6] Geis T. and Ebner J. 2001, Flow structures inside a rotor-stator cavity, International Journal of Rotating Machinery, Vol. 7, No. 4, pp. 285-300.
[7] Miner S.M. 2001, 3-D viscous flow analysis of a mixed flow pump impeller, International Journal of Rotating Machinery, Vol. 7,
[8] Miyauchi S., Horiguchi H. and Fukutomi J.-I., Takahashi A., 2004, Optimization of meridional flow channel design of pumpimpeller, International Journal of Rotating Machinery, Vol. 10, pp. 115–119.
[9] Nagnostopoulos J. 2006. CFD analysis and design effects in a radial pump impeller, WSEAS Transactions on Fluid Mechanics, Vol.1, pp. 763-769.
[10] Patel K., Ramakrishnan N. 2006, CFD analysis of mixed flow pump, International ANSYS Conference Proceedings.

