Volume: 5 Issue: IV Month of publication: April 2017

DOI: http://doi.org/10.22214/ijraset.2017.4267

Volume 5 Issue IV, April 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Internal Flow Analysis of Submersible Pump Impeller using Computational Fluid Dynamics

K. Manikandan [1], L. Periyannan[2]
[1] PG Scholar, [2]AP, Department of Mechanical Engineering
Mahendra Engineering College Namakkal, TN, India

Abstract: Submersible pump has been playing an important role in industrial and house hold applications. A submersible pump (or electric submersible pump (ESP)) is a device which has a hermetically sealed motor close-coupled to the pump body. The whole assembly is sub merged in the fluid to be pumped. The submersible pumps used in ESP installations are multistage centrifugal pumps operating in a vertical position. The liquids, after being subjected to great centrifugal forces caused by the high rotational speed of the impeller, lose their kinetic energy in the diffuser where a conversion of kinetic to pressure energy takes place. And thus in this pump impeller plays an important role by which the efficiency has been calculated.In an impeller the design parameters such as number of blades, blade angles, diameter of the impeller, width of the blades are the important parameter to be considered because which affects the performance of the pump. And so here we made an analysis on the impeller by changing the outlet blade angle from the existing blade. The analysis is made by using Computational Fluid Dynamics (CFD) software by which the hydraulic efficiency has been calculated. The results are obtained from the Computational Fluid Dynamics (CFD) it has been calculated that by increasing the outlet blade angle by 5^0 the hydraulic efficiency of the impeller has been increased by 9.85% from existing impeller model which has the hydraulic efficiency of 73.5%. It has been evident that by increasing the blade angle the hydraulic efficiency is increased. For each impeller, the flow pattern and the pressure distribution in the blade passages are calculated and finally the head-capacity curves are compared and discussed.
Keywords: ANSYS, Hard Chrome, Thermal Boundary

I. INTRODUCTION

A. *Submersible Pumps*

The submersible pumps used in Electric submersible pump installations are multistage centrifugal pumps operating in a vertical position. Produced liquids, after being subjected to great centrifugal forces caused by the high rotational speed of the impeller, lose their kinetic energy in the diffuser where a conversion of kinetic to pressure energy takes place. This is the main operational mechanism of radial and mixed flow pumps.

B. *The Centrifugal Impeller*

The fluid enters at centre of the wheel, turns through a right angle and, as it moves outwards radially, is subjected to centrifugal force resulting in an increase in its static pressure. Rotational and radial components of velocity are imparted to the fluid. The corresponding outlet velocity pressure may then be partially converted into static pressure.

II. IMPELLER MODEL AND ITS SPECIFICATIONS

The impeller model and their specification is given by the Hari Industries is given below:

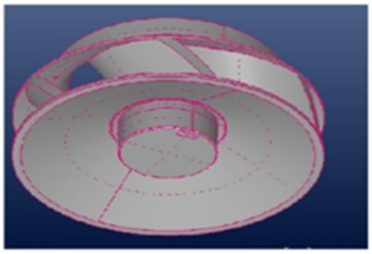


Fig 1 Impeller Model

Volume 5 Issue IV, April 2017
ISSN: 2321-9653

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

TABLE I Impeller parameters:

IMPELLER INLET(D_i)	75mm
IMPELLER OUTLET(D_O)	105mm
BLADE NUMBER	6
INLET BLADE ANGLE(β_i)	69^0
OUTLET BLADE ANGLE(β_O)	49^0
BLADE THICKNESS(t)	1.25mm
BLADE INLET HEIGHT(L_i)	21mm
BLADE OUTLET HEIGHT(L_O)	16mm

As for the existing impeller model, an design optimisation has been planned by changing their number of blades(n), Inlet blade angle(α) and Outlet blade angle (β) using CFD software in terms of Trial and Error method and kept the other specifications as to be constants.

III. METHODOLOGY

In all of these approaches the same basic procedure is followed.

A. During preprocessing

B. The geometry (physical bounds) of the problem is defined.

C. The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform.

D. The physical modeling is defined.

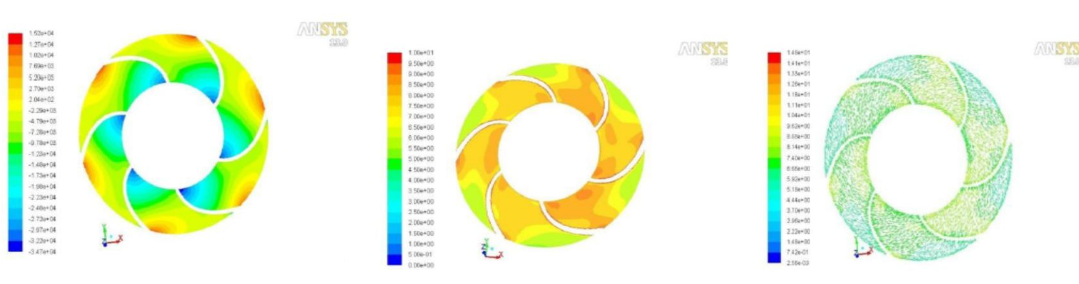
E. Boundary conditions are defined. This involves specifying the fluid behaviour and properties at the boundaries of the problem. For transient problems, the initial conditions are also defined.

F. The simulation is started and the equations are solved iteratively as a steady-state or transient.

G. Finally a postprocessor is used for the analysis and visualization of the resulting solution.

IV. SOFTWARE ANALYSIS MODEL

Analysis for Existing Impeller - Static pressure (Pascals), relative velocity magnitude (m/s), Velocity Vector



Static pressure (Pascals) Relative velocity magnitude (m/s) Velocity Vector

Fig 2 Analysis for Existing

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

V. RESULT AND DISCUSSIONS

In this graph we can find that at 540 outlet blade angle the discharge and head will be maximum when compared to the outlet blade angle 440 and 490. Hence it is conclude that at 540 angle the head and discharge will be higher than the other angles 440 and 490. Various results are shown in the following figures.

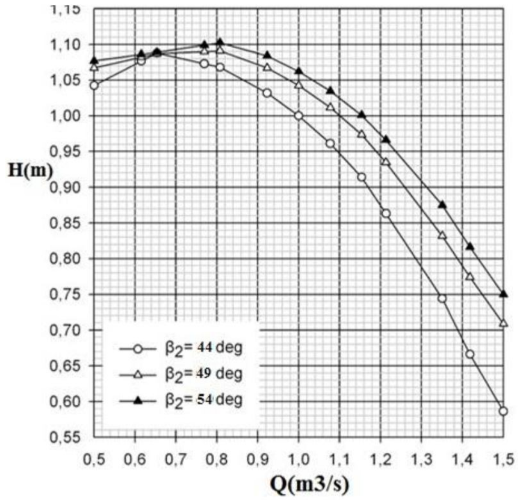


Fig 3 Head & Discharge curve

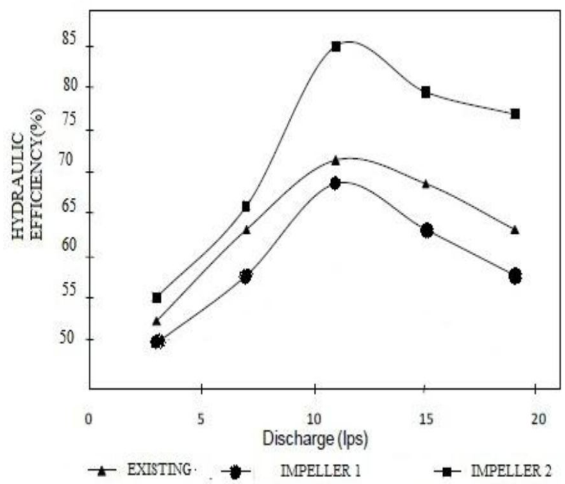


Fig 4 Hydraulic Efficiency & Discharge

VI. CONCLUSION

Here the impeller model of a submersible pump has been analysed mathematically and analytically. In mathematical analysis the existing impeller model have an hydraulic efficiency of 65.3% and furthur the same model has been analysed made by using Computational Fluid Dynamics (CFD) software then it has been obtained that the analytical hydraulic efficiency is 73.5%. From the above results it has been conclude that analytical results of the existing impeller is higher than the mathematical results. And further the same existing impeller has been optimum at the outlet blade angle region which is decreased to 5^0 and increased to 5^0 angle. The existing impeller has an outlet blade angle of 49^0 and the modified angle is 44^0 for model 1 and 54^0 for model 2. For these two model an analysis is done by using same Computational Fluid Dynamics (CFD) software and it shows the results of hydraulic efficiency as 67.12% for model 1 and 83.35% for model 2. It has been seen that by increasing the blade angle the hydraulic efficiency of the impeller gets incresed but the pressure head range of the impeller is decreased when compared to the existing model and optimum model 1. Thus by increasing the outlet blade angle the hydraulic efficiency has been increased by 9.8% when compared to the existing model.

Volume 5 Issue IV, April 2017
ISSN: 2321-9653

REFERENCES

[1] Manivannan, "Computational fluid dynamics analysis of a mixed flow pump impeller" International journel of science and technology Vol. 2, No. 6, 2010, pp. 200-206

[2] Khalid. S. Rababa, The Effect of Blades Number and Shape on the Operating Characteristics of Groundwater Centrifugal Pumps . European Journal of Scientific Research ISSN 1450-216X Vol.52 No.2 (2011), pp.243-251

[3] 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, 2, 75-83

[4] LIU Houlin*, WANG Yong, YUAN Shouqi, TAN Minggao, and WANG Kai, Effects of Blade Number on Characteristics of Centrifugal Pumps, Chinese Journal of Mechanical Engineering Vol. 23,2011.

[5] Christopher Earls Brennen "Hydrodynamics of Pumps" by OPEN © Concepts NREC 1994.

[6] Yunus A.Cengel and John M.Cimbala " Fluid Mechanics "

[7] D. Eckardt, "Detailed flow investigations within a high-speed centrifugal compressor impeller", ASME Journal of Fluids Engineering, vol. 98, pp. 390-402, 1976

[8] M. W Johnson, and J. Moore, "The development of wake flow in a centrifugal impeller", ASME Journal of Engineering for Power, vol.102, pp. 382-390, 1980.

[9] Kjork, and L. Lofdahl, "Hot-wire measurement inside a centrifugal impeller", ASME Journal of Fluids Engineering, vol. 111, pp. 363-368, 1989.