International Journal of Applied Engineering Research ISSN 0973-4562 Volume 10, Number 18 (2015) pp 38972-38976

The Effect of Impeller Meridional Shape and Vane Curvature Modifications in a Mixed Flow Submersible Borewell Pump

C. Murugesan,
Vice President-Design and Development, Aquasub Engineering, Thudiyalur Post, Coimbatore, Tamilnadu, India cmurugesan@aquagroup. in

R. Rudramoorthy,
Principal, PSG College of Technology, Coimbatore, Tamilnadu, India principal@psgtech. edu

Abstract—Modifications were done in the impeller meridional shape and vane curvature of a mixed flow submersible borewell pump to reduce impeller losses and improve the efficiency. The first part of this paper gives the numerical simulation of existing pump performance by using Ansys Fluent. In the second part the meridional shape of the impeller and vane curvature were thoroughly analysed and modified using the inverse design code TURBOdesign1.In the third part the numerical simulation of the modified pump was made. The results showed that the pump efficiency had improved by 4%.The flow losses were reduced in the impeller passage after the modification. A prototype pump was made and experimental test was conducted. The experimental results also confirmed the numerical simulation results.

Key words: Impeller meridional shape, Vane curvature, Inverse design, Flow loss, Mixing loss, Numerical Simulation, Mixed flow submersible bore well pump.

Introduction

Submersible borewell pump was invented in the year 1916 [1] in Ukraine by Armais Arutunoff and a US patent was obtained in 1927. He started the first factory to manufacture them in the name of REDA(Russian Electrical Dynamo of Arutunoff) in 1928 at Bartlesville, Oklahoma, USA. These submerible pumps were used in oil wells to pump crude oil. Oil filled motors were used in these pumpsets. Later the same design concept was evolved into wet type winding motor filled with water and fitted with multistage centrifugal pumps suitable to borewells. These pumpsets were used for pumping water from borewells.

The mixed flow submersible pumps are used to pump water from higher depths of borewells. These pumps are used in the borewells dug in the alluvial soil formation mainly in the delta regions of the rivers. The population of these pumps are increasing in the last two decades all over the world.The submersible pump of mixed flow type is shown in Fig1.The inlet bracket(1) is used to guide the water entry to the first stage mixed flow impeller(2). The rotating blade system in the impeller imparts energy to the water and water comes out with high velocity in a conical spiral pattern. The diffuser(3) blade passages collect the water, diffuses its velocity and direct the water to the next stage impeller. The velocity of the water at the second stage is kept same as that of the first stage, but the pressure energy is higher which is gained at the first impeller. This process repeats in all the subsequent stages and the energy is added in all the stages by the work done at the impeller passages. The discharge casing(4) is at the end of the stages. It collects the water and directs it to the column pipe(5) which is transferring the water to the ground level. Also column pipe is supporting the entire pumpset i.e pump coupled with the motor and the axial down thrust caused by the pump impellers during running.

In the last decade, various studies were conducted on the impeller and volute interaction of centrifugal radial flow pumps and the results were published. The numerical simulation of the dynamic effects due to impeller-volute interaction in a centrifugal pump was studied by Jose

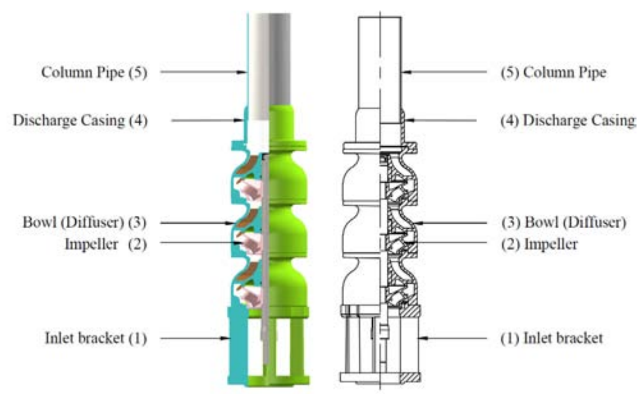


Fig. 1. Submersible pump Mixed flow type

Gonzalez,et.al[2].The head and flow distribution within the volute of a centrifugal pump in comparison with the characteristics of the impeller without casing was given by P.Hergt et.al [3].Also Akhras.A et.al [4] discussed the flow rate influence on the interaction of a radial pump impeller and the diffuser.Jianjun Feng,et.al[5] described the numerical investigation on pressure fluctuation for different configurations of vane diffuser pumps. The virtual performance experiment of a centrifugal pump was given by Zhang Shujia,et.al[6].The CFD application and validation for a turbo machinery design system was described by Anderson,M.R.et.al[7].

The numerical analysis of the unsteady flow in the near tongue region in a volute-type centrifugal pump for different operation point was presented by Raul Barrio,et.al [8]. Perez.J,et.al [9] have done experiments in a two stage radial flow pump and compared it with numerical simulations.B.Jafarzadeh, et.al [10] have studied the flow simulation of turbulent fluid flow in a low specific speed centrifugal pump.Paul Cooper,et.al [11] have analysed the complex internal flows in the centrifugal pumps. Stefania Della Gatte, et.al [12] have done CFD for the assessment of axial thrust balance in radial flow multistage pumps. Milan Sedlar,et.al[13] have numerically analysed the mixed-flow pump with volute. The design criteria for the suppression of secondary flows in the mixed flow impeller was given by Zanganeh et.al [14]. Goto ,A et.al [15] had explained the design criteria for mixed flow diffusers using inverse design method and CFD.

Pump Parameters

The specifications of the test pump is given below:

TABLE 1 Test pump parameters

Symbol	Parameter	Value	Unit
Hn	Head (Nominal) of 3 Stage at BEP	24 (8 x 3)	m
Qn	Discharge Rate (Nominal) at BEP	0.008	m^3/s
N	Speed of Rotation	2880	rpm
Ns	Specific Speed (metric) $n\,Q_n^{0.5}\,H_n^{-0.75}$ (min-1, m^3/s ,m)	54.15	-
D_1	Impeller Diameter, Inlet (Hub/Shroud)	27/64	mm
D_2	Impeller Diameter, Outlet (Hub/Shroud)	93/94	mm
b_2	Impeller Width, Outlet	13	mm
b_1	Impeller Width, Inlet	21.5	mm
Z_i	No. of Impeller Vanes	7	-
T	Vane Thickness	3	mm
D_3	Diffuser Inlet Diameter, Mean	108	mm
D_5	Diffuser Outlet Diameter,Mean	50	mm
Z_d	No. of Diffuser Vanes	8	-
P_n	Power Rating of 3 stage (Pump Input)	3.35 (1.11 x 3)	kW
BEP	Best Efficiency Point	-	-
ηp	Pump Efficiency	-	%

Numerical Simulation

The flow domain was modeled and meshed with optimum grid size for the pump. Inlet pressure, outlet pressure and speed of rotation were given as initial boundary conditions to the flow volume domain. After the convergence of the solution the surface integrals for the pressure difference across inlet and outlet was calculated and total head was arrived. The surface integral of mass flow rate at outlet gives the discharge rate. The impeller wall surfaces and blade surfaces where the momentum was imparted to the fluid were selected. The surface integral of the momentum in these surfaces were calculated to arrive at the torque imparted by the impeller and the input power to the pump. For the given total head and resulting discharge rate, the output was calculated. The values of grid size and selection of turbulence models play vital role to achieve close matching with experimental data. Sensitivity analysis is required before carrying out these simulations. The flow volume was split into 4 domains as inlet duct, outlet duct, diffuser and impeller. The grid dependency study was done in the three stages with the grid cells ranging from 0.25 million to 1 million for the entire mass flow region from 0.5 to 1.5 times the nominal flow. 0.75 million grid cells were selected considering accuracy and computation time. The influence of turbulence models on the predictions was investigated with the selected grid cells. The investigation was done with k-Epsilon Standard, k-Omega Standard and k-Omega SST. It was observed that the k-Omega SST model gives accurate predictions in higher flow rate conditions also. The k -Omega SST model was chosen finally for further analysis.

A. Analysis of pump performance using Numerical results:

To study the pump performance, the numerical results of the existing pump was analysed for nominal discharge rate condition. The velocity contours in the meridional view of the impeller and "Blade to Blade" view of the impeller spanwise from hub to shroud was studied.In the meridional view the velocity was not following the shroud wall as shown in Fig 2 due to sudden change in the meridional shape of the shroud. Also it was found from the blade to blade view of the impeller in Fig 3 that the flow velocity was uniform up to 70% of span. From 70% to 100% of span the flow velocities are not uniform and recirculation takes place near suction side of the blades. These two phenomena reduces the impeller efficiency and it was estimated as 88% by Ansys CFD- Post[16].

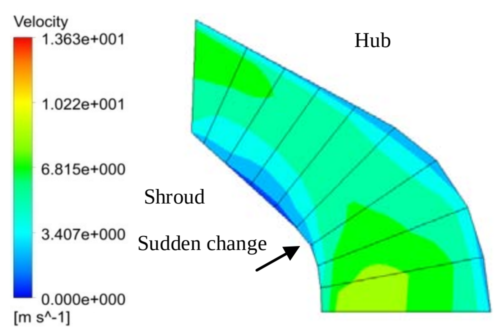
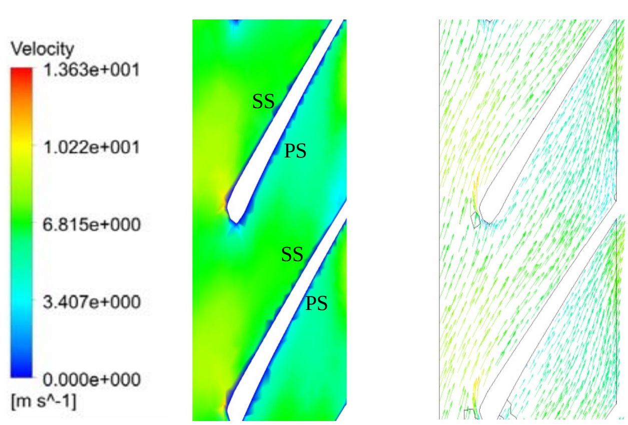
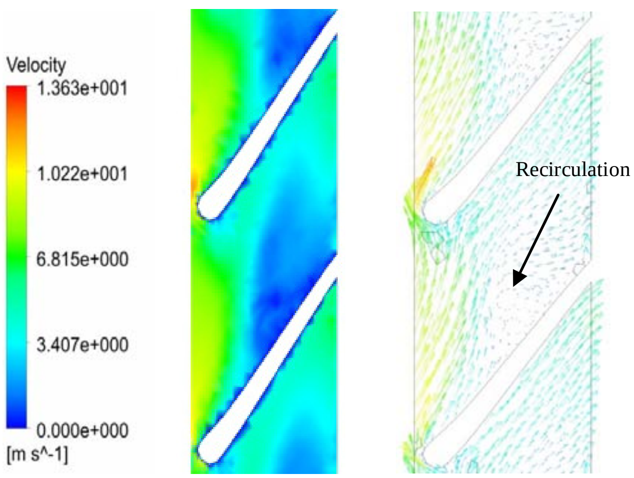


Fig. 2 Velocity contours in Meridional view of the existing impeller

To reduce the flow separation from the shroud wall in the meridional view of the impeller, the shroud shape was modified with smooth curvature change without sudden turn as shown in Fig 6.Also to improve the flow separations found at 90% of the spanwise blade to blade of impeller, blade shape was modified using inverse design method using the concept of positive staking in impeller leading edge , front loading in the shroud and rear loading in the hub as shown in Fig 4 & 5.This was suggested by Zangeneh et.al[14] and Akiro Goto et.al [15] as the inverse design method for mixed flow pump impellers.

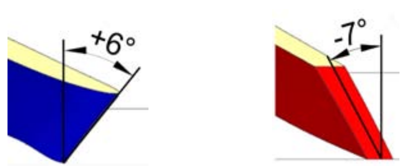


At 50% span of Blade to Blade View



At 90% span of Blade to Blade View

PS - Pressure Side of Blade SS - Suction Side of Blade
Fig 3 Spanwise Blade to Blade View of existing impeller



Modified Impeller Existing Impeller

Fig 4 Impeller stacking

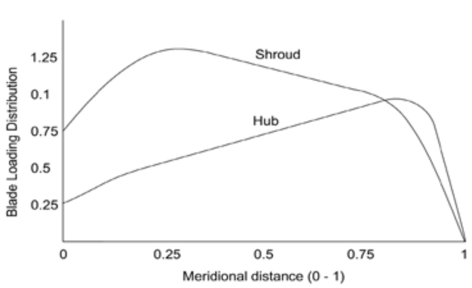


Fig 5 Blade Loading Definition of Modified Impeller

Numerical simulation was done for the modified impeller with existing bowl. The results showed considerable increase in the impeller efficiency from 88% to 93% and flow separations found in the meridional view and spanwise blade to blade view has been reduced as shown in Fig 7 and Fig 8. Because of these two modifications the overall pump efficiency has been increased to 74% from 70%.

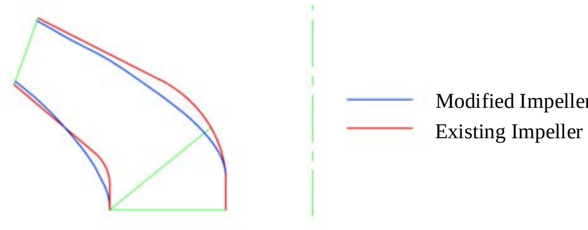


Fig 6 Meridional view of the impeller

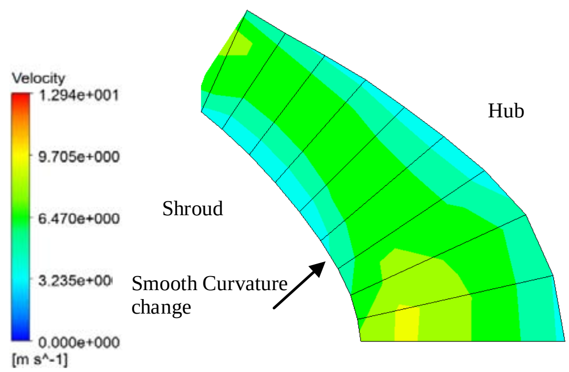
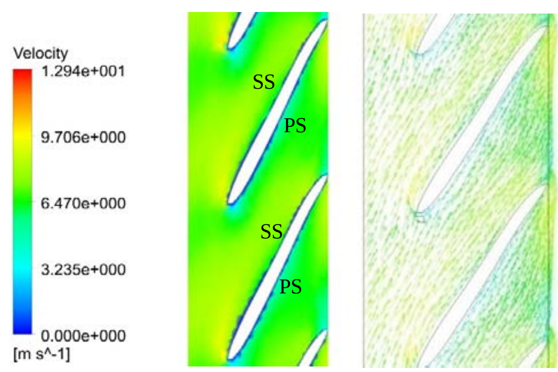
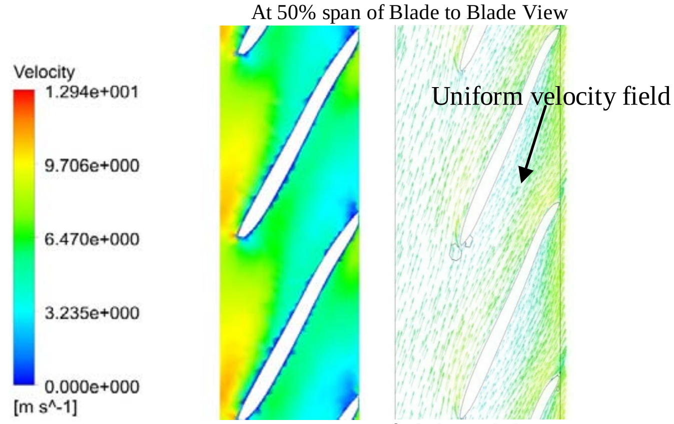


Fig 7 Velocity contours in Meridional view of the Modified impeller



At 50% span of Blade to Blade View



At 90% span of Blade to Blade View
Fig 8 Spanwise Blade to Blade View of Modified impeller

Experimental Performance Comparison

Based on the numerical results prototype impeller was made and experimental performance test was conducted. Existing and modified impeller were tested in the experimental test set-up as per the ISO pump testing standard ISO 9906.The test rig scheme is shown in Fig. 9.

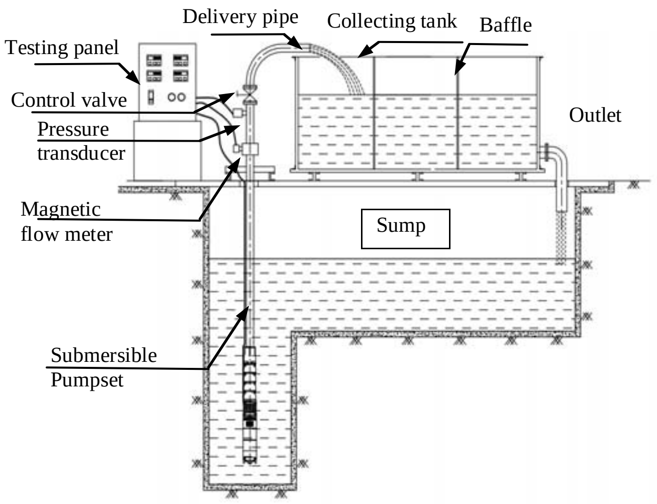


Fig 9 Experimental Test Setup as per ISO:9906

The pressure measurements were taken for different discharge rates of the pump. The input to the pump was measured in kW and the output was the product of head (pressure difference across inlet and outlet) and mass flow rate. The modified impeller pump efficiency showed 4% increase from the existing design as predicted by numerical simulation as shown in Fig 10 at BEP of the pump.

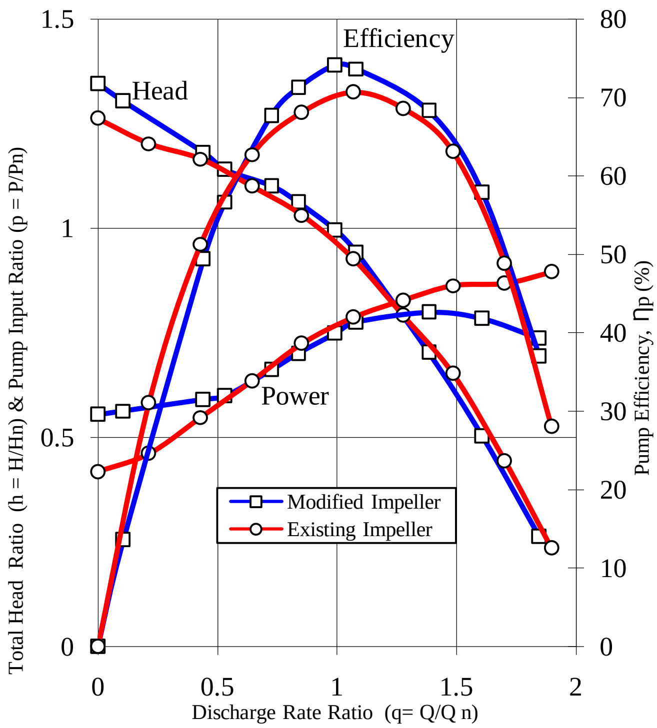


Fig 10 Experimental Performance Curves

Conclusion:

The smooth curvature of the shroud surface of the mixed flow impeller play a vital role in maintaining the uniform velocity distribution in the meridional view. Sudden change in the shroud should be avoided during design stage to minimise the flow separation in the impeller passage.

Optimising the blade shape using inverse design method helps to suppress the secondary flows across the blade to blade view. Optimum blade loading that is loading the front portion of the shroud and rear portion of the hub along with positive stacking at trailing edge of the impeller blades reduces the flow disturbance at impeller exit.

When the flow separations were reduced in the impeller passage and uniform velocity field was maintained, the pump efficiency increases at BEP of the pump characteristics.

Acknowledgement

The authors thank Dr.K.M.Srinivasan,Dr.P.R.Thiyagarajan, Dr.K.Mayilsamy and Dr.T.Prabhu of PSG College of Technology for their valuable inputs during this study. Authors thank the management of M/s Aquasub Engineering, P.Ramesh and G.Prasath for their support in carrying out the numerical and experimental study.

Reference

1. Home page on esp pump [online] Available : http:// www.esppump.com.
2. Jose G, Joaquin F, Eduardo B and Carlos S, Numerical Simulation of the Dynamic Effects Due to Impeller – Volute Interaction in a Centrifugal Pump, *J Fluids Eng,Trans ASME*,124 (2002) 348-355.
3. Peter H, Stephan M and Bernd S, The Flow and Head Distribution with in the Volute of a Centrifugal Pump in Comparison with the Characteristics of the Impeller Without Casing,*J Comput Appl Mech* ,Miskolc University Press,Hungary, 5(2004), 275-285.
4. Akhras A, El Hajem E, Morel R and Champagne J Y, The Flow Rate Influence on the Interaction of a Radial Pump Impeller and the Diffuser, *Int J Rotating Mach,Taylor & Francis*,10(2004), 309–317.
5. Jianjun F, Friedrich-Karl B and Hans J D, Numerical Investigation on Pressure Fluctuations for Different Configurations of Vaned Diffuser Pumps, *IJRM, Hindawi* ,2007.
6. Zhang S, Zhu B,Hu Qo and Li X, Virtual Performance Experiment of a Centrifugal Pump, *Proc 16th Int Conf on Artificial Reality and Teleexistence(* ICAT 06), IEEE,2006.

7. Anderson M R, Gu, Fahua, MacLeod P D ,Application and Validation of CFD in a Turbo Machinery Design System, *Int Mech Engg Congress and R & D Expo,ASME*.

8. Barrio R, Fernandez J, Blanco E and Parrondo J,Estimation of radial load in centrifugal pumps using computational fluid dynamics, *Eur J Mech B-Fluid* 30(2011)316-324.

9. Perez J, Chiva S, Segala W, Morales R,Negrao C, Julia E and Hernandez L,Performance Analysis of Flow in a Impeller-Diffuser Centrifugal Pumps using CFD: Simulation and Experimental Data Comparisons, *EECMV*,ECCOMAS CFD 2010.

10. Jafarzadeh B,Hajari A,Alishahi M M and Akbari M H, The flow simulation of a low - specific- speed high- speed centrifugal pump, *Appl Math Model*, *Elsevier*, 35,(2011), 242–249.

11. Paul C, Edward G and Timothy L, Computational Fluid Dynamics Analysis of Complex Integral Flows in Centrifugal Pumps, *Proc Int Pump Users Sym*, (Turbomachinery Laboratory, Texas A & M University System), 1994.

12. Stefania D G, Simone S, Paolo A and Laura B, CFD Study For Assessment of Axial Thrust Balance in Centrifugal Multistage Pumps, *Proc Int Conf Fluid Flow Tech, CMFF06*.

13. Milan S, Numerical Investigation of Flow in Mixed-Flow Pump with Volute, *Task'u*, 5 (2001), 595–602.

14. Zangeneh,M., Goto A, 1998 Hydrodynamic Design of Pump Diffuser using Inverse Design Method and CFD,*ASME Paper*,*FEDSM98-4854* 1998.

15. Akira GOTO, Kosuke ASHIHARA, Takaki SAKURAI, Sumio SAITO, Pump Design System based on Inverse Design Method and its Application to Development of Diffuser Pump Series,*Proceedings of ASME 1999*.

16. Ansys, "Fluent theory Guide", 2012.