ANALYSIS AND INVESTIGATION OF CENTRIFUGAL PUMP IMPELLERS USING CFD

P.ArunKumar
Assistant Professor
Department of Mechanical Engineering
Karpagam University, Coimbatore
Tamil Nadu, India

P.Dhachinamoorthi
Assistant Professor
Department of Mechanical Engineering
Karpagam University, Coimbatore
Tamil Nadu, India.

K.Saravanakumar,
Assistant Professor,
Department of Production Engineering,
P.S.G College of Technology, Coimbatore,
Tamil Nadu, India

S.Venkatesh
Assistant Professor,
Department of Mechanical Engineering
Sri Eshwar College of Engineering,
Coimbatore, Tamil Nadu, India

Abstract

This project investigates the study of complex internal flows in centrifugal pump impellers with the aid of Computational Fluid Dynamics software thus facilitating the design of pumps. Here three different types of Pump impellers had been taken (one with three straight blades, one with four straight blades and the other one with four twisted blade).The pump specifications considered for investigation are discharge and speed. These specifications have been varied to perform a comparative study of these pump impellers. The impeller was modeled in Pro Engineering and the blade to blade plane of the impellers was taken for the detailed study purpose because the flow occurs through this passage only. The blade to blade plane is modeled in GAMBIT software and the flow analysis is carried out using FLUENT software. Thus the valid results regarding the velocity distribution and pressure distributions were predicted and the performance of those pumps had been compared from the computational results

Keywords: CFD, Pump Impeller,efficiency,velocity

I. INTRODUCTION

All standard The hydraulic machines which convert the mechanical energy in to hydraulic energy are called Pumps. If the mechanical energy is converted into pressure energy by means of centrifugal forces acting on the fluid, then the hydraulic machine is called as centrifugal Pump. The centrifugal pump was developed in Europe in the late 1600’s and was seen in the united states in the early 1800’s. Its wide spread use, however has occurred only in the last seventy five years. Research and development has resulted in both improved performance and new materials of construction that have greatly expanded its field of applicability. It is common today to find efficiencies of above 90%for the case of large pumps.

Casing receives liquid at higher velocity from the impeller and converts it into pressure energy. The impeller often discharges directly into the volute, which is a spiral-shaped flow passage usually of circular or trapezoidal cross section. Its cross section increases gradually around the impeller periphery, starting from the tongue and then ending in the volute throat. The volute tongue directs the total flow, collected from around the impeller, through the throat to the pump diffuser. A diffuser may or may not exist between the volute throat and the exit flange, depending on the available space and the flow velocities.

Computational Fluid Dynamics (CFD) is a computer-based tool for simulating the behavior of systems involving fluid flow, heat transfer and other related physical processes. It works by solving the equations of fluid flow over a region of interest, with specified (known) conditions on the boundary of the region. CFD has grown from a mathematical curiosity to become an essential tool in almost every branch of fluid dynamics, from aerospace propulsion to weather prediction. This CFD is commonly accepted as referring to the broad topic encompassing the numerical solution, by computational methods, of the governing equations which describe the fluid flow, the set of Navier-Strokes equations, continuity and any additional conservation equations, for example energy (for heat transfer).

II. LITERATURE REVIEW

Extensive research work in the area of impeller has been going on over the last few decades in order to improve their performance. The flow process is highly complex in the water pump impellers and it can be predicted well with the aid of CFD and thus facilitating the design of pumps. A CFD approach seems a logical way to have a detailed look at the flow behavior and to predict the regions of separation with a high degree of accuracy. Thus CFD is an important tool for pump designers.

Oh J.S, RO H.S and Goto. A [1]Oh and Ro used a compressible time marching method, a traditional Simple method, and a commercial program of CFX-TASC flow to simulate flow pattern through a water pump and compared the difference between among these methods in predicting the pumps performance. Goto presented a comparison between the measured and computed exit-flow fields of a mixed flow impeller with various tip clearances, including the shrouded and un-shrouded impellers, and confirmed the applicability of the incompressible version of the three-dimensional Navier-stokes code developed by Dawes for a mixed flow centrifugal pump.Zhou Weidong, Ng and his colleagues [2] Zhou and Ng and their colleagues also developed a three – dimensional time-marching, incompressible Navier-stokes solver using the pseudo compressibility technique to study the flow field through a mixed-flow water-pump impeller. The applicability of the original code was validated by comparing it With many published experimental and computational results.Kaupert, potts, Tsukamoto [3] Kaupert and his colleagues, Potss, and Sun and Tsukamoto studied pump off-design performance using the commercial software CFX-TASC flow, FLUENT, and STARCD, respectively. Although these researchers predicted reverse flow in the impeller shroud region at small flow rates numerically, some contradictions still existed. Kaupert‘s experiments showed the simultaneous appearance of shroud-side reverse flow at the impeller inlet and outlet but his CFD results failed to predict the numerical outlet – reverse flows. Sun and Tsukamoto validated the predicted results of the head-flow curves, diffuser inlet pressure distribution, and impeller radial forces by revealing the experimental data over the entire flow range, and they predicted back flow at small rates, but they did not show an exact back- flow pattern along the impeller outlet. These researches clearly show the necessities of the off-design performance in pumps are necessary.Miguel Asuaje, Farid Bakir, Sma¨ıne Kouidri, Frank Kenyery [4]A 3D-CFD simulation of the impeller and volute of a centrifugal pump has been performed using CFX codes. The pump has a specific speed of 32 (metric units) and an outside impeller diameter of 400 mm. First, a 3D flow simulation for the impeller with a structured grid is presented. A sensitivity analysis regarding grid quality and turbulence models were also performed. The final impeller model obtained was used for a 3D quasi-unsteady flow simulation of the impeller-volute stage. A procedure for designing the volute, the nonstructural grid generation in the volute, and the interface flow passage between the impeller and volute are discussed. This flow simulation was carried out for several impeller blades and volute tongue relative positions. As a result, velocity and pressure field were calculated for different flow rates, allowing to obtain the radial thrust on the pump shaft.

III. MATERIALS AND METHODOLOGY

In this research three types of centrifugal pump impellers considered. For designing the impellers eleven important parameters are considered. Each parameter values are shown in table -1. Here three cases are considered. Volumetric and pump gross efficiency was calculated with help of equations 1 and 2 for each case of impeller design parameters. The three types of impeller diagrams are shown in figure 1,2 and 3. The figure 4 shows the shape of twisted vane. The table -2 gives the volumetric and gross efficiency values for three cases of impellers. From these results case-3 gives the better result compare to another two cases. Finally it was validated by computational fluid dynamics technique.

Volumetric efficiency =1/ (1+aN_s-0.66) ---------------- (1)
Pump gross Efficiency = $_v$ x $_H$ x $_m$ -------------------------------- (2)

Table -1 Design Parameters

S.no	Design parameters	Values		
		Case I	Case II	Case III
1	Discharge (Q)	70 m^3/hr	90.87 m^3/hr	114 m^3/hr
2	Head (H)	30 m	27 m	29 m
3	Speed (N)	1400 rpm	1400 rpm	1400 rpm
4	Specific speed (N_s)	55.52 rpm	70 rpm	70.4 rpm
5	Inlet to outlet ratio (a)	0.65	0.65	0.65
6	Normalized diameter (D_1)	0.102 m	0.11m	0.0906 m
7	Mechanical efficiency ($_m$)	91%	90%	94%
8	Velocity at inlet	9.38 m/s	9.89 m/s	8.97m/s
9	Vane angle	19°	17°	28.8°
10	Number of vanes (Z)	3	4	4
11	Hydraulic efficiency ($_H$)	0.93	92.5%	93.6%

Table -2 Efficiency Values.

Efficiency	Case I	Case II	Case III
Volumetric efficiency ($_v$)	96%	94%	96.2%
Pump gross efficiency ($_g$)	81.2%	77%	83%

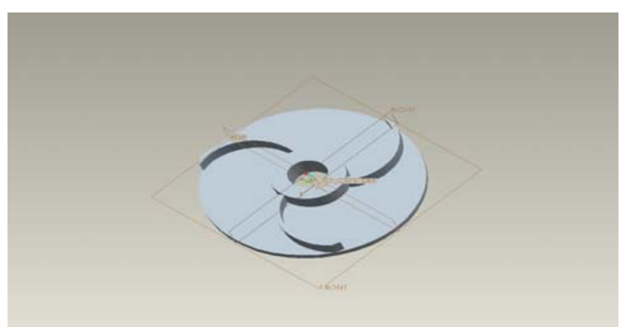

Figure 1. Impeller with 3 vanes

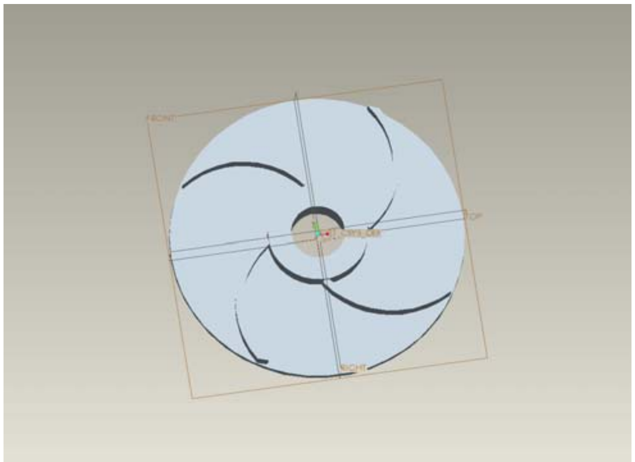

Figure 2 Impeller with 4 vanes

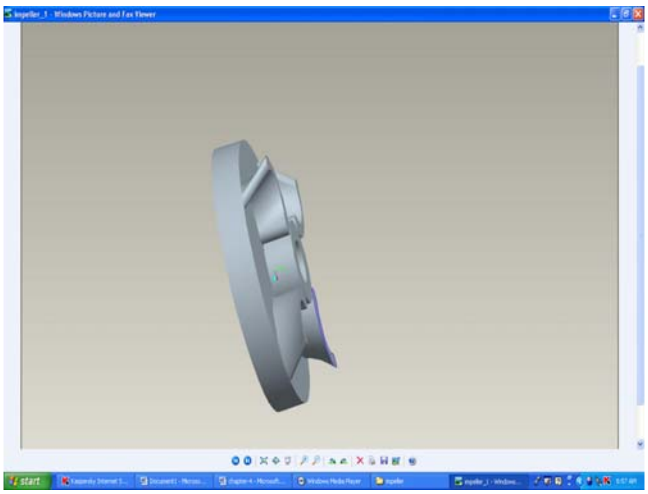

Figure 3 Impeller with 4 twisted vanes

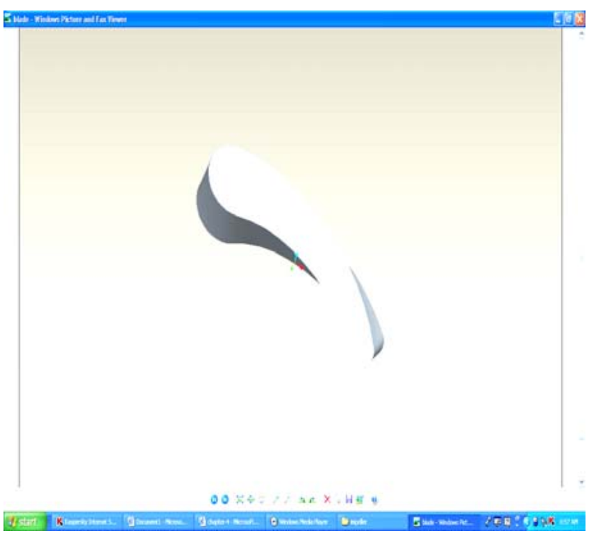

Figure 4 shape of the twisted vane

IV. RESULTS AND DISCUSSIONS

A. Comparison Of Twisted Vanes And Flat Blades (Case1)

In the first case the Impeller 2 and Impeller 3 are compared to analyze their performance. The impeller 2 consists of four flat backward curved vanes and the impeller 3 consists of four twisted vanes. In this case we can compare by changing the speed of the pumps. Here the rpm selected for analyze is 2000rpm is selected. So both the impellers are designed at the speed 2000 rpm. Here the blade to blade plane of the impeller had been taken for analyze purpose. Here the comparison is between their vector velocity distribution and their contour pressure distributions. The figure 6 shows the meshed model of the vane. The figure 7 and 8 gives the velocity distribution results and figure 9 and 10 gives the pressure distribution results.

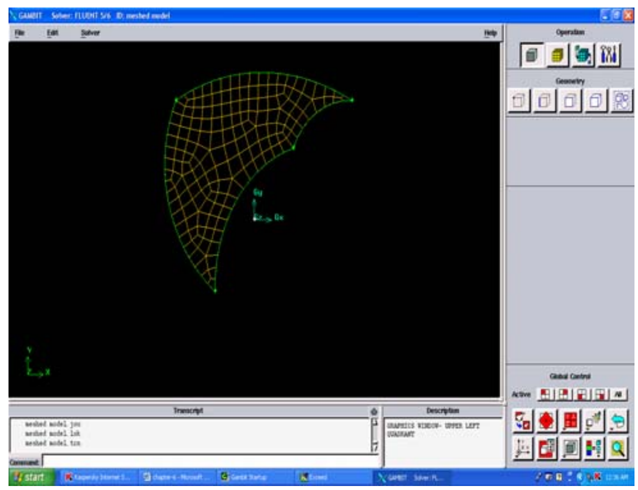

Figure 6. Meshed model of the blade to blade plane of the impeller

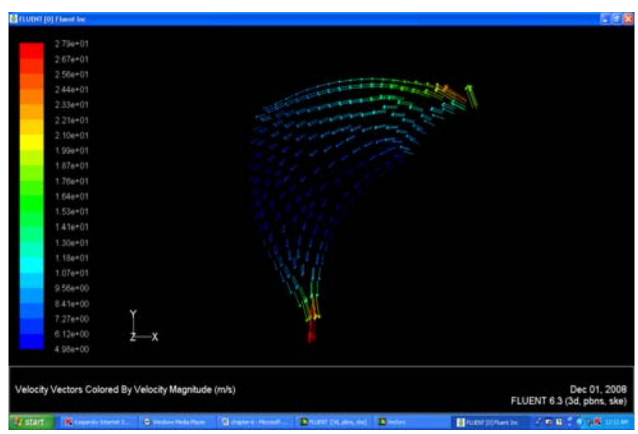

Figure 7. Velocity distribution for Impeller 2 when N=2000rpm

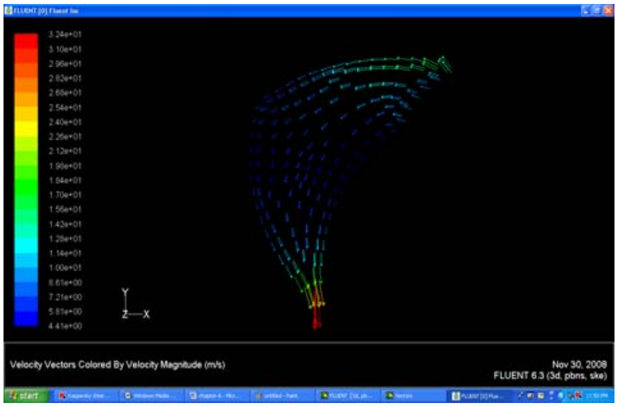

Figure 8. Velocity distribution for impeller 3 when N= 2000 rpm

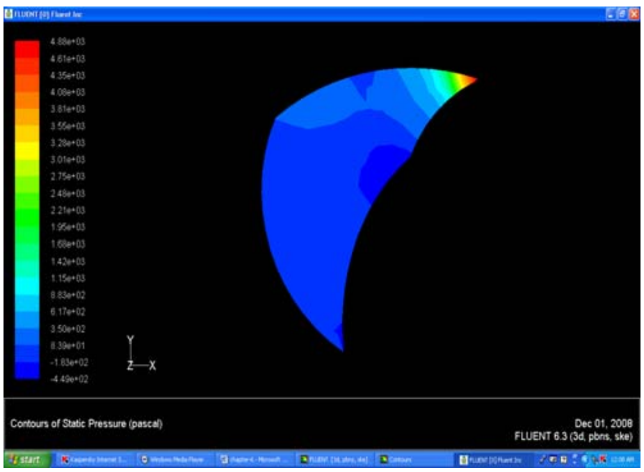

Figure 9. Pressure distribution for impeller 2 for N=2000rpm

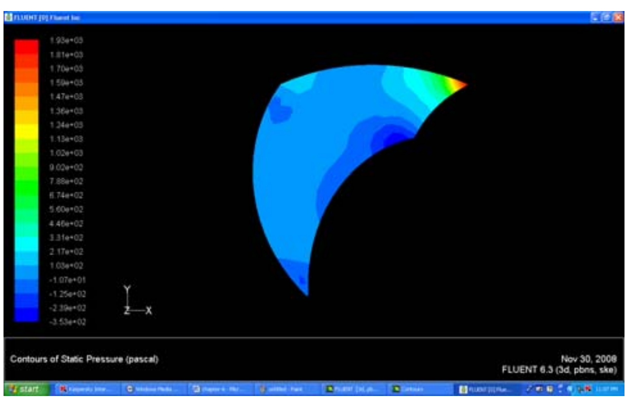

Figure 10. Pressure distribution for impeller 3 for N=2000rpm

B. Results for case1

From the blade to blade plane velocity distribution diagram and from the pressure contours it is clearly found that the flow was much smoother in both the cases. The pressure at the exit is higher for the twisted vanes when compared to that of the flat vanes. The pressure in both the cases rises linearly. Normally it has high pressure on the pressure surface than on the suction surface. As the flow passage through the blade to blade plane is small it shows a slight increase in pressure. By comparing the velocity distribution for both the impellers it is found that the performance of twisted blade to blade plane is higher than those of the flat one.

TABLE 3 RESULT VALUES FOR CASE 1

S.N o	Impeller type	Speed	Velocity at the inlet m/s	Velocity at the outlet m/s	Pressure at the outlet Pascal
1	4 flat type	2000	22.45	10.52	350
2	4twisted type	2000	19.8	7.85	560

From the values in the above table it is found that the twisted vane has high pressure outlet value 560 Pascal when compared to that of flat type blades. Here the values of velocity drop and the value of pressure increases in the outlet side.

C. Comparison Of Impeller 1 And Impeller 2 (Case2)

The Impeller 1 consists of 3 flat backward curved vanes and the Impeller 2 consists of 4 flat backward curved vanes. Both these impellers are analyzed in order to compare their performance. Here the discharge had been altered for the two pumps. For the Pump Impeller 1 the operating speed is 1400 rpm and the two discharges are 70m3/hr and 45m3/hr. For the Pump impeller 2 also the operating speed is same but the discharge is 60m3/hr and 90m3/hr. As the comparison is between the same types of blades that is flat backward curved vane and also as they have the same speed therefore here it is enough to compare their velocity vector distribution. The figure 11 and 12 ,13 and 14 gives the velocity distribution results.

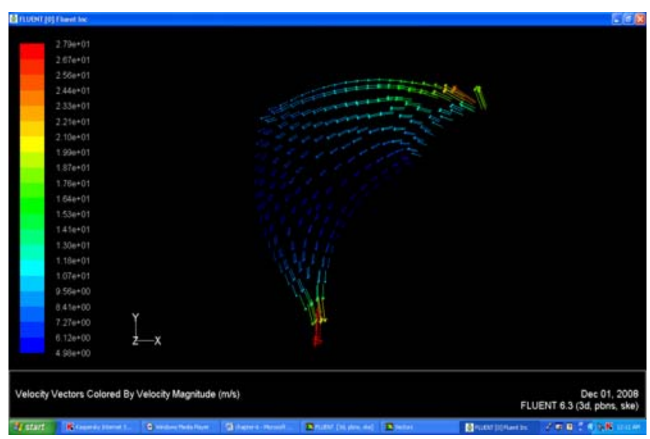

Figure 11. Velocity distribution for impeller 1 for N=1400rpm and Q=70m3/hr

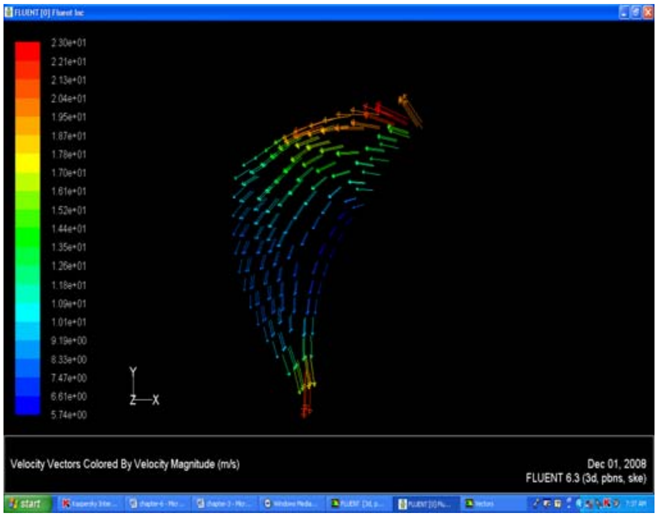

Figure 12.Velocity distribution for impeller1 for N=1400rpm and Q=45m3/hr

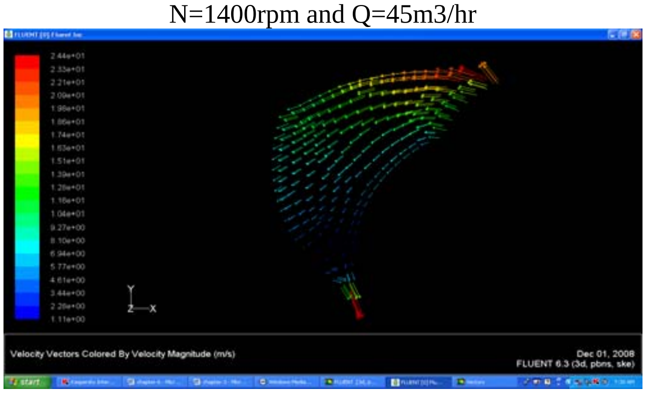

Figure 13. Velocity distribution for impeller2 for N=1400rpm and Q= 60 m3/hr

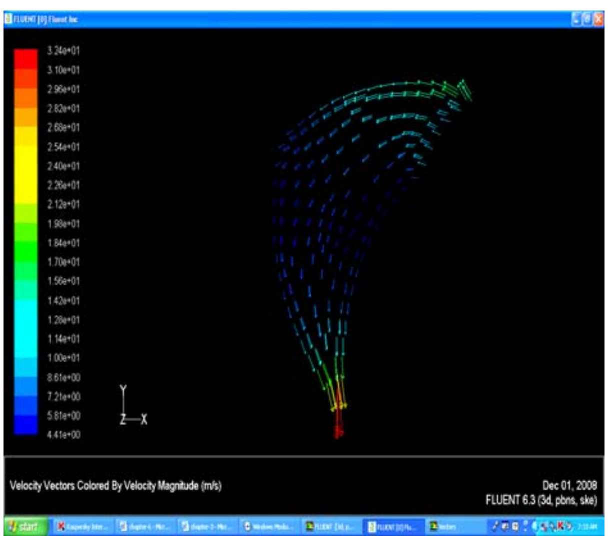

Figure 14. Velocity distribution for impeller 2 for N=1400rpm and Q=90m3/hr

D. Results For Case 2

From the velocity vector distribution the recirculation area had been checked. It is clear that the flow is smooth in both the cases but there will be a slight disturbance in the flow at the suction side because the suction area is somewhat narrow passage here the waters enter with the high speed as the pressure drops in the suction side. When the flow rate is 90m3/hr for the impeller 2 the flow is smooth. But when it is 60m3/hr there is a small change in the flow direction and it may leads to recirculation of fluid. Similarly in the case of impeller 1 also the flow is smooth with the discharge of 70m3/hr when compared to the flow rate of 45 m3/hr. Here there is no problem related with recirculation and the flows through the pumps are smooth

TABLE 4. RESULT VALUES FOR CASE 2

S.No	Impeller vanes	Discharge m3/hr	Speed rpm	Velocity at the inlet m/s	Velocity at the outlet m/s
1	3	70	1400	13.3	9
2	3	45	1400	14.4	8.52
3	4	60	1400	16.3	5.77
4	4	90	1400	12.8	6.21

So when comparing the velocity values it is found that the drop in velocity is very high for the 4 vane impeller so as a result the 4 vane impeller is better than three vane impeller.

V. CONCLUSION

The impeller was designed and modeled to analyze its performance. The blade to blade plane of the impeller had been taken for analysis. The analysis was carried out in Fluent (Computational fluid dynamics). The velocity and pressure distribution in the blade to-blade plane was studied. Design of an impeller was carried out by considering the Head, Discharge and the speed of the pump. Here the performance of three pumps had been studied by changing their specifications. The results obtained from CFD are used to compare their performance. It was found that the predicted results for pump3 is better than those of the pump1 and pump2, which suggested that the efficiency of the pump3 will be higher than the other two pumps. From the second case it was found that when the inflow rate is 20% of the design flow rate, no deviation in flow pattern but if the flow rate drops or increase by (30-40%) then the flow pattern deviates slightly and there is a chance for the reverse flow to take place. Here there is no problem related with recirculation and the flows through the pumps are smooth.

REFERENCES

[1] Application of the Full Cavitations Model to Pumps and Inducers MAHESH M. ATHAVALE*, H. Y. LI, YU JIANG, and ASHOK K. SINGHAL CFD Research Corporation, 215 Wynn Dr., Huntsville, Alabama, 35805, USA International Journal of Rotating Machinery, 8(1): 45±56, 2002 Copyright # 2002 Taylor & Francis 1023-621X/02

[2] Performance Analysis for Submersible Pump –A Numerical Approach Thitipong Charoenkijtavee, Vejapong Juttijudata1, Panumas Arundachawat Bangkok 10900, Thailand ANSCSE10 pp 404.

[3] Effect of Reynolds-number and surface roughness on the efficiency of centrifugal pumps J.F. Gülich, Sulzer Pumps, Winterthur and Ecole Polytechnique, Lausanne, Switzerland ASME Journal of Fluids Engineering 125 (2003) 4, 670-679

[4] Dynamic characteristics and leakage performance of liquid annular seals in centrifugal pumps Eskild Storteig Department of Marine Engineering Norwegian University of Science and Technology NTNU Submitted 99

[5] Multiphase Pumping: Achievements and Perspectives J. Falcimaigne1, J. Brac1, Y. Charron1, P. Pagnier1 and R. Vilagines1Oil & Gas Science and Technology – Rev. IFP, Vol. 57 (2002), No. 1, pp. 99-107Copyright © 2002, Editions Technip

[6] Proceedings of the Ninth NRC/ASME Symposium on Valves, Pumps and In service Testing Held at L'Enfant Plaza Hotel Washington, DC July 17-19, 2006

[7] Optimum values of design variables versus specific speed for centrifugal pumps, H.W Oh and M. K Chung,Taejon, Korea A05098 IMechE 1999

[8] Jacek wojtusik, 2003, "Application of multimodels to modeling of centrifugal pumps behaviour", AI-Meth, Pp 339-344.

[9] Shukla.S.N and Kshirsagar.J.T. 2003, "Numerical experiments on a centrifugal pump", Indian pumps. Pp 21-30.

[10] Church and Lal, 1973, "Centrifugal pumps and blowers" Metropolitan book Co. Pvt. Limited.

[11] Lazarkiewiz "Design of Pumps and impellers"

[12] Herbert addision, "Centrifugal and other rotadynamic pumps".

[13] 3D flow simulation in an axial flow pump using computational fluid dynamic method, B.C Bhayal , R & D division , INDIAN pumps 2002.

[14] Effect of Reynolds-number and surface roughness on the efficiency of centrifugal pumps J.F. Gülich, Sulzer Pumps, Winterthur and Ecole Polytechnique, Lausanne, Switzerland ASME Journal of Fluids Engineering 125 (2003) 4, 670-679

[15] Yagnesh Sharma.N, and KaranthK.V., 2003, "CFD analysis of a turbulent steady flow through a volute casing of a centrifugal pump for static pressure distribution", ISME, Pp MD-036.

AUTHORS PROFILE

P.Arunkumar received his Bachelor of Engineering Degree in Mechanical Engineering from Government College of Technology, Anna University, Chennai, India in 2006, Master of Engineering Degree in Engineering Design from Anna University of Technology, Coimbatore, India in 2010. He is currently working as an Assistant Professor in Department of Mechanical Engineering, Karpagam University, Coimbatore, India. He has 5 years of experience in teaching and 4 years of experience in industry. He has published 1 paper in International and 3 Papers in National Conference.

K.Saravanakumar received his Bachelor of Engineering Degree in Mechanical Engineering from Hindusthan College of Engineering and Technology, Anna University, Chennai, India in 2008, Master of Engineering Degree in Production Engineering from PSG College of Technology, Anna University, Coimbatore, India in 2010. He is currently working as an Assistant Professor in Department of Production Engineering, PSG College of Technology, Coimbatore, India. He has 4 years of academic experience in Teaching. He has published 2 papers in International Journals and 5 papers National Conference.

P.Dhachinamoorthi received his Bachelor of Engineering Degree in Mechanical Engineering from Bannari Amman Institute of Technology, Anna University, Chennai, India in 2006, Master of Engineering Degree in Engineering Design from Karpagam University, Coimbatore, India in 2011. He is currently as working as an Assistant Professor in Department of Mechanical Engineering, Karpagam University, Coimbatore. He has 7 years of experience in teaching. He published 5 papers in National Conference.

S.Venkatesh received his Bachelor of Engineering Degree in Mechanical Engineering from Anna University, Chennai, and M.E Engineering Design from Anna University Chennai. Presently working as an Assistant Professor in Mechanical Engineering Department, Sri Eshwar College of engineering, Coimbatore. He has got 5 years of teaching experience and 3 years industrial experience and also he published 3 international journals and 2 national conferences. His research interests include CFD, optimization, and design of experiment.