Available online at www.sciencedirect.com

SciVerse ScienceDirect

Procedia Engineering 51 (2013) 715 – 720

Procedia Engineering

www.elsevier.com/locate/procedia

Chemical, Civil and Mechanical Engineering Tracks of 3rd Nirma University International Conference (NUiCONE 2012)

CFD for centrifugal pumps: a review of the state-of-the-art

S R Shah[a], S V Jain[b*], R N Patel[b], V J Lakhera[b]

[a]*Dharmsinh Desai University, Nadiad-387001, India*
[b]*Institute of Technology, Nirma University, Ahmedabad-382481, India*

Abstract

The flow analysis inside the centrifugal pump is highly complex mainly due to 3D flow structure involving turbulence, secondary flow, cavitation and unsteadiness. In recent years, a growing availability of computational resources and progress in the accuracy of numerical methods brought turbo machinery Computational Fluid Dynamics (CFD) methods from pure research work into the competitive industrial markets. The critical review of CFD analysis of centrifugal pumps along with the future scopes for further improvement is presented in this paper. CFD technique has been applied by the researchers to carry out different investigations on centrifugal pumps viz. performance prediction at design and off-design conditions, parametric study, cavitation analysis, diffuser pump analysis, performance of pump running in turbine mode etc. Unsteady Reynolds-averaged Navier–Stokes equations together with two equation k-ε turbulence model were found to be appropriate for CFD analysis of centrifugal pump. Volute flow study and impeller-volute interaction appeared as an interesting research fields for the further improvement in the pump performance. The most active areas of research and development are the analysis of two phase flow, pump handling non-Newtonian fluids and fluid–structure interaction.

Keywords: Centrifugal pumps; CFD; flow analysis; numerical simulation; review.

1. Introduction

Centrifugal pump is a most common pump used in industries, agriculture and domestic applications. Its impeller design demands a detailed understanding of the internal flow at rated and part load operating conditions [1]. For the cost-effective design of pumps it is very crucial to predict their performance in advance before manufacturing them, which requires understanding of the flow behavior in different parts of the pump. Experimental model testing is one of the solutions for prediction of performance but it is tedious, time consuming and costly. Conversely, theoretical approach merely gives a value; but it is unable to determine the root cause for the poor performance [2]. In the recent years, CFD started to play a key role for the prediction of the flow through pumps and turbines having successfully contributed to the enhancement of their design [3].

The application of CFD in the design of water pumps and turbines started about 30 years ago. The first steps coincided with the introduction of the finite element method into CFD and were characterized by simplified Quasi-3D Euler solutions and fully 3D potential flow solutions. Over the years the complexity continuously increased in stages: via 3D Euler solutions, to steady Reynolds Averaged Navier-Stokes (RANS) simulations of single blade passages using finite volume methods, extending to steady simulations of whole machines, until today unsteady RANS equations are solved with advanced turbulence models. The most active areas of research and development are now concerned with including the effects of 2-phase flow and fluid–structure interaction [4, 5].

* Corresponding author. Tel.: 09998623087; fax:+02717-241917.
E-mail address: sanjay.jain@nirmauni.ac.in

doi:10.1016/j.proeng.2013.01.102

A growing availability of computer power and a progress in accuracy of numerical methods, brought turbomachinery CFD methods from pure research work into the competitive industrial markets [6]. Many softwares are available in the market for numerical analysis of turbo-machines viz. Fluent (UK and US), CFX (UK and Canada), Fidap (US), Polyflow (Belgium), Phoenix (UK), Star CD (UK), Flow 3d (US), ESI/CFDRC (US), SCRYU (Japan) and more [7]. Recently, Kaupert et al. [8], Potts and Newton [9] and Sun and Tsukamoto [10] studied pump off-design performance using the commercial software CFX-TASCflow, FLUENT and STARCD respectively.

Normally, the CFD codes provide three calculation methods for the analysis of turbomachinery flows: the Multiple Reference Frame (MRF), the Mixing Plane and the Sliding Mesh. The first two methods are basically steady flow methods. In MRF method, the rotor is kept at a fixed position and the governing equations for rotor are solved in a rotating reference frame, *including Coriolis and centrifugal forces,* while for the stator are solved in an absolute reference frame [11]. CFD has been widely used in low specific speed and medium specific speed mixed flow pumps by Takamara and Goto [12] and Goto [13]; and in radial flow pump by Sedlar and Mensik [14].

In this paper, the critical review of the CFD applications in centrifugal pumpsis presented. The advantages and limitations of CFD technique are described. The current trends and the future scopes for the flow analysis of centrifugal pump using CFD as a numerical simulation tool are discussed.

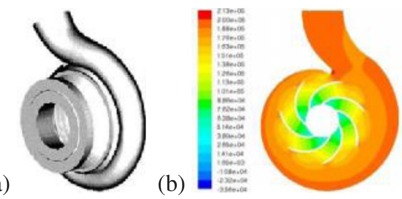
2. Literature review

Pump designers are continually being challenged to provide machines that operate more efficiently, quietly and reliably at lower cost. Many investigators have applied CFD as a numerical simulation tool to carry out different investigations on centrifugal pumps. This section describes the research work carried out by the researchers in the centrifugal pumps by CFD approach.

2.1. Performance prediction at different operating conditions

Centrifugal pumps are widely used in many applications, so the pump system may be required to operate over a wide flow range in different applications. The most previous numerical studies were focused on the design or near-design state of pumps. Few efforts were made to study the off-design performance of pumps, where the performance of pump deteriorates [15]. With the aid of the CFD approach, the complex internal flows through the different components of pump can be studied at different operating conditions which help in improvement in the performance at off-design conditions.

Mentzoset al. [16] carried out a numerical simulation of the internal flow in a backward curve vaned centrifugal pump. The MRF approach used to take into account the impeller-volute interaction was completely failed, due to its fixed coupling formulation. However, its use was recommended for basic understanding of the flow at various operating points. The transient analysis was suggested as a real tool for understanding of the interaction between impeller and spiral casing. Three-dimensional computational model of centrifugal pump is shown in Fig. 1(a).



(a) (b)

Fig. 1.(a)Three-dimensional computational model of centrifugal pump and (b)static pressure contours in the pump.

Mentzos et al. [17] simulated the flow through the impeller of centrifugal pump using finite-volume method along with a structured grid system for the solution of the discretized governing equations. The CFD technique was applied to predict the flow patterns, pressure distribution and head-capacity curve. It was reported that, although the grid size was not adequate to investigate the local boundary layer variables, global ones were well captured. The proposed approach was advocated for the basic understanding of the flow at various operating points.

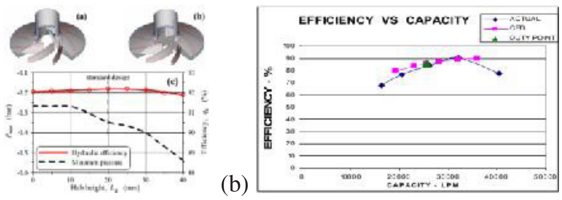
Shah et al. [18] carried out steady state simulation of 200 m^3/hr capacity centrifugal pump using using RANS equations. The non-uniformities were observed in different parts of the pump at off-design conditions which resulted in the decrease in efficiency. The k-ω SST turbulence model provided better results compared to RNG k-ε model. The operating characteristic curves predicted by the numerical simulation were compared with the results of model testing and were found in good agreement. The static pressure contours in the pump at rated discharge are shown in Fig. 1(b).

2.2. Parametric study

CFD helps in prediction of flow behavior in different parts of the hydraulic machines before actually manufacturing them. In case of modification of existing systems, the modifications can be incorporated in numerical model and their effects can be predicted before implementing them. CFD analysis helps in studying the effects of various parameters, independently as well as by forming the non-dimensional groups, on pump performance.

Bacharoudis et al. [19] analyzed the performance of pump by varying the outlet blade angles by keeping the same outlet diameter. The numerical simulation of 3-D, incompressible Navier-Stokes equations was carried out with a commercial CFD finite-volume code. At nominal capacity, when the outlet blade angle was increased from 20° to 50°, the head was increased by more than 6% but the hydraulic efficiency was reduced by 4.5%. However, at high flow rates, the increase of the outlet blade angle caused a significant improvement of the hydraulic efficiency.

Anagnostopoulos [20] developed numerical model for the simulation of the 3-D turbulent flow in centrifugal pump impeller to solve the RANS equations. The impeller geometry was represented by a number of controllable design variables, providing the capability of modifying the impeller shape and testing different configurations. The results of such parametric studies showed that, a remarkable gain in hydraulic efficiency may be achieved by optimizing the impeller geometry. Modified shroud geometry and its effect on head and efficiency are shown in Fig. 2(a).



(a) (b)

Fig 2.(a) Modified shroud geometry and its effect on head and efficiency and (b)efficiency versus capacity curve.

Patel and Ramakrishnan [21] numerically studied the effects of changing hub curve profile and stator angle in mixed flow pump at duty point and at part load. The analysis concluded that: (i) the nature of head & power versus capacity curves obtained was similar to that of standard mixed flow pump (ii) pump efficiency was predicted within + 5% range at duty point. However, more variation was observed at off-design conditions and (iii) efficiency was improved by 1% after matching stator angle and changing hub curve profile. The efficiency versus capacity curves, *actual and predicted by CFD analysis*, are shown in Fig. 2(b).

2.3. Cavitation analysis

Cavitation may occur in different regions of the pump when local pressure goes below the vapour pressure correspond to fluid temperature. The mechanism of cavitation erosion has been studied for more than a hundred years, but until now there has been no general theory of cavitation erosion damage to analytically calculate cavitation erosion rate in impellers of centrifugal pumps or to evaluate erosion intensity at the pump design stage.

Medvitz et al. [22] used multi-phase CFD method to analyze centrifugal pump performance under cavitating conditions. The homogeneous two phase RANS equations were used wherein mixture momentum and volume continuity equations were solved along with vapor volume fraction. Performance trends ofpartial discharge and blade cavitation, *including breakdown*, were observed and compared qualitatively with experimental measurements.

Nohmiet al. [23] studied the cavitation flow in a low specific speed centrifugal pump with compressible air-vapor-liquid two-phase medium (TE model) and constant enthalpy vaporization (CEV) model. The study revealed that, at the high flow rate cavitation bubbles appear at the leading edge on pressure side and the head drops gradually. The TE model was able to predict the gradual head drop but the computations were found to be unstable; whereas, CEV model was unable to predict the gradual head drop. In both the codes, further modification was recommended to achieve stable and accurate results.

Caridad et al. [24] carried out numerical analysis in a centrifugal pump impeller of submersible pump conveying an air–water mixture, which was similar to cavitating flow.A sensibility analysis with regard to the gas-void fraction and the bubble diameter was performed. The variations in impeller head and relative flow angle at the outlet were presented as a function of liquid flow rate and phase distribution within the impeller. It was found that, larger bubble diameter lead to larger head experimented by the impeller. The numerical results and diffuser losses showed excellent agreement with the experimental results.

2.4. Investigations on interacting componenets

The relative movement between impeller and volute generates an unsteady interaction which affects not only the overall pump performance but is also responsible for pressure fluctuations. Pressure fluctuations interact with the volute casing and give rise to dynamic effects (mainly unsteady forces) over the mechanical parts, which are one of the most important sources of vibration and hydraulic noise. Gonzalez et al. [25] showed the capability of a numerical simulation in capturing the dynamic and unsteady flow effects inside a centrifugal pump due to impeller-volute interaction. Viscous Navier-Stokes equations along with sliding mesh technique was applied to consider the impeller-volute interaction. The amplitude of the fluctuating pressure field at the blade passing frequency was successfully captured by the model for a wide range of operating flow rates. Both experiments and numerical prediction showed the presence of a spatial fluctuation pattern at the blade passing frequency as a function of the flow rate.Pressure fluctuations in the volute wall is shown in Fig. 3(a).

Wang and Tsukamoto [26] developed numerical method for more realistic prediction of pressure fluctuations due to rotor-stator interaction in a diffuser pump by considering the change in operating point of the pump. Fig. 4 shows velocity vectors around channel 1 and 2 of impeller blades under rotating stall. The pressure fluctuations were predicted in 2-D unsteady incompressible flow using a vortex method, in which vortices shed from solid boundary were determined based on the momentum equations. It was reported that the pressure in the diffuser passage fluctuates with the basic frequency of the impeller blade passing frequency.

2.5. Axial thrust in centrifugal pumps

One of the most challenging aspects in horizontal shaft pump is evaluation of the axial thrust acting on the rotating shaft which is affected by pump characteristics, working conditions and internal pressure fields. Solving this problem is simple for single stage pumps while several complications arise for multistage pumps. Salvadori et al. [27] carried out CFD simulation of horizontal multistage pump and evaluated the contribution of each component to the axial load by investigation of its internal flow and pressure field. The methodology was suggested for proper dimensioning of multistage pump thrust bearings which guaranteed the reliability of the pump. The importance of leakage flows in shroud chambers and balancing drums for axial balancing of pump was also discussed. The proposed technique was able to consider the effects of pump operating and wear conditions on axial thrust.

Gatta et al. [28] presented a methodology to predict the residual axial thrust in a multistage pump and discussed the different variables influencing the axial thrust. A mixing plane technique was used to study the interaction between rotating and stationary parts which has considered the effects of different incidences and inlet velocity distortions. It was mentioned that the side chambers give a great contribution to the axial thrust and their pressure distribution is highly influenced by leakage mass flow and local rotating speed inside the cavity.

2.6. Non-newtonian fluid handling pumps

Pumps are widely used in transporting slurries [29] from one point to another in mining industry, chemical industry, metallurgical operations, coal industry etc. In typical slurries, solid particles have a range of diameters and concentration depending on type of slurry for a particular application. The presence of solid particles affects the hydrodynamic and erosion performance of the pump.

Pagalthivarthi et al. [30] simulated dense slurry flow through centrifugal pump casing using the Eulerian multiphase model in FLUENT 6.1. First order upwind scheme was considered for the discretization of momentum, k and ε terms and a mixture property based k-ε turbulence model was used for modeling turbulence. The effects of pump flow rate, tongue curvature, casing width, inlet concentration of the particles was considered on wall stress distribution and velocities along the wall. Analysis concluded that solid concentration and solid wall shear stress increase monotonically from the upstream of the tongue to the downstream of the belly region.

Yu et al. [31] investigated the effects of impeller geometry on the performance of a centrifugal blood pump both experimentally and computationally. Four impeller designs were tested at the operating point (i.e. 2000 rpm), using blood analog as the working fluid. It was found that the stress levels within the blade passages were generally below the threshold level of 150 N/m2 for extensive erythrocyte damage to occur. Some localized regions near the leading edge of the blades were observed where the stress levels were 60% above the threshold level.

2.7. Pump running in turbine mode

A small pump running as turbine can be much more economically incorporated in micro hydropower plants considering various advantages associated with pumps like low initial and maintenance cost, available for a wide range of heads and flows, spare parts are easily available, easy installation etc. [32]. Many investigators have carried out CFD analysis of pump as turbine (PAT). Natanasabapathi and Kshirsagar [33] investigated the performance of PAT using numerical approach. The 3-D computational model was created using Pro-E solid modeller and transferred to CFX as shown in Fig. 3(b). Initially, the simulations were carried out with unstructured mesh in which, the head drop across the turbine matched with experimental values but deviations were observed in efficiency at discharges away from BEP. Later, two rings of structured grid were introduced in between casing and runner which showed a good trend with the experimental results.

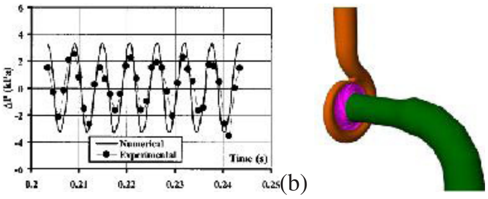
(a)  (b)

Fig.3.(a)Pressure fluctuations in the volute wall in angular position opposite to the tongue and (b) computational model of PAT.

Fernandez et al. [34] carried out 3-D simulation of a PAT using a sliding mesh technique which made it possible to account for the effect of blade–tongue interactions on the local flow. The comparison of numerical and experimental data is shown in Fig. 5(a). A maximum relative error of 9 percent was obtained for the total head and about 5 percent for the average static pressure around the impeller periphery.

Barrio et al. [35] carried out internal flow analysis of a centrifugal pump in pump and turbine modes using the commercial code Fluent. The simulations were carried out by solving the full unsteady Reynolds-averaged Navier–Stokes (URANS) equations with the finite volume method for the flow rates between 20 to 160 per cent. The numerical results were compared with the experimental measurements which showed typical differences in the range 3–5 percent. In turbine mode, internal recirculation in the impeller was found for low and high flow rates.

2.8. Mini/micro pump analysis

For the turbo-pump being the key machine for liquid transportation, its further development is always desirable. Impeller diameter between 5 mm and 50 mm is defined as mini pump. Liu et al. [36] carried out experimental and numerical studies on impeller-geometry of mini turbo-pump. The law of similitude was observed for the pump characteristics in the range of Reynolds number larger than 1.0×10^5. The effect of tip clearance was found to be attenuated by the impeller geometry such as larger outlet blade angle. It was concluded that, numerical 3-D flow analysis based on RANS equations with k-ω turbulence model may be reasonably applicable to study the hydraulic performance of mini impellers.

Tsui and Lu [37] analyzed the unsteady flow field prevailing in the valve less micro pump by using both the CFD and the lumped-system method. The moving membrane was modeled by imposing a reciprocating velocity boundary condition. In the multidimensional simulation, the Navier–Stokes equations were solved using a finite volume method suitable for the use of unstructured grids. It was reported that the variation of the flow rate ratio is quite different in the lumped-system analysis compared with the multidimensional calculations, due to negligence of inertial effect.

3. Conclusions

Flow analysis of centrifugal pump is often a challenging task as it requires critical analysis of highly complex flow which is turbulent and three dimensional in nature and having rapidly changing curvature of flow passage. CFD approach has been extensively used in centrifugal pumps as numerical simulation tool for performance prediction at design and off-design conditions, parametric study, cavitation analysis, analysis of interaction effects in different components, prediction of axial thrust, study of pump performance in turbine mode, diffuser pump analysis etc. URANS equations together with two equation k-ε turbulence model were found to be appropriate to get a reasonable estimation of the general performance of the centrifugal pump, *from an engineering point of view*, with typical errors below 10 percent compared with experimental data. Impeller and diffuser flows have been studied extensively and volute flow study has appeared as an interesting research field for further improvement of the pump performance. The most active areas of research and development are the analysis of 2-phase flow (cavitation and slurry flow), pump handling non-Newtonian fluids and fluid–structure interaction. CFD approach provides many advantages compared to other approaches; however due to the empirical nature of solution technique validation with experimental results is usually recommended. The findings of the present study may be useful to the researchers working in the similar area.

References

[1] Usha, P.,Syamsundar, C.,2010. "Computational analysis on performance of a centrifugal pump impeller," Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power. Chennai, India, paper#TM-07.

[2] Patel, K., Satanee, M., 2006. "New development of high head Francis turbine at jyoti ltd. for small hydro power plant," Proceedings of Himalayan Small Hydropower Summit. Dehradun, India, paper#13.

[3] Croba, D.,Kueny, J., 1996. Numerical Calculation of 2D Unsteady Flow in Centrifugal Pumps: Impeller and Volute Interaction.International Journal for Numerical Methods in Fluids22,p. 467.

[4] Keck, H., Sick, M., 2008. Thirty years of numerical flow simulation in hydraulic turbomachines. ActaMech 201, p. 211.

[5]Keck, H., Weiss, T., Michler, W., Sick, M., 2007. "Recent developments in the dynamic analysis of water turbines,"Proceedings of the 2nd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, Timisoara, Romania, pp. 9-20.

[6] Pascoa, J., Mendes, A., Gato, L., 2009.A Fast Iterative Inverse Method for Turbomachinery Blade Design.Mechanics Research Communications36(5), p. 537.

[7] http://www.cfdreview.com.

[8] Kaupert, K., Holbein, P., Staubli, T., 1996. A First Analysis of Flow Field Hysteresis in a Pump Impeller.Journal ofFluids Engineering 118, p. 685.

[9] Potts, I., Newton, T., 1998. "Use of commercial CFD pack-age to predict shut-off behavior of model centrifugal pump: an appraisal,"IMechE Seminar.

[10] Sun, J., Tsukamoto, H. 2001. Off-Design Performance Prediction for Diffuser Pumps.Journal of Power and Energy, Proceedings of I. Mech. E A215, p. 191.

[11] Dick, E.,Vierendeels, J.,Serbruyns, S.,Voorde, J., 2001. Performance Prediction of Centrifugal Pumps with CFD-Tools. Task Quarterly 5 (4), p. 579.

[12]Takemura, T., Goto,A., 1996.Experimental and Numerical Study of Three-Dimensional Flows in a Mixed-Flow Pump Stage.JournalofTurbomachinery118 (3), p. 552.

[13]Goto, A., 1997. Predictionof Diffuser Pump Performance Using 3D Viscous Stage Calculation.Journal of FluidsEngineering.

[14]SedlarMandMensik P 1999 Investigation of rotor/stator interaction influence on flow fields in radial flow pumps Proceedings of 3rd European Conference on Turbomachinery, IMechE,pp.1017–1025

[15] Hedi, L., Hatem, K.,Ridha, Z., 2010."Numerical flow simulation in a centrifugal pump," International Renewable Energy Congress. Sousse, Tunisia. pp. 300-304.

[16] Mentzos, M.,Filios, A.,Margaris, P.,Papanikas, D.,2004. "A numerical simulation of the impeller-volute interaction in a centrifugal pump," Proceedings of International Conference from Scientific Computing to Computational Engineering. Athens, pp. 1-7.

[17] Mentzos, M., Filios, A., Margaris, P., Papanikas, D., 2005. "CFD predictions of flow through a centrifugal pump impeller," Proceedings of International Conf. Experiments/Process/System Modelling/ Simulation/Optimization. Athens, pp. 1-8.

[18] Shah, S., Jain, S.,Lakhera, V., 2010. "CFD based flow analysis of centrifugal pump," Proceedings ofInternational Conferenceon Fluid Mechanics and Fluid Power. Chennai, India, paper#TM08.

[19]Bacharoudis, E.,Filios, A.,Mentzos, M.,Margaris, D., 2008.Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle. The Open Mechanical Engineering Journal 2, p. 75.

[20] Anagnostopoulos, J., 2006. CFD Analysis and Design Effects in a Radial Pump Impeller. WSEAS Transactions on Fluid Mechanics 1 (7), p. 763.

[21] Patel K.,Ramakrishnan, N., "CFD analysis of mixed flow pump."

[22] Medvitz, R., Kunz, R., Boger, D., Adam, J.,Yocum, A., Pauley, L., 2002. Performance Analysis of Cavitating Flow in Centrifugal Pumps Using Multiphase CFD. Journal of Fluid Engineering 124, p. 377.

[23] Nohmi, M.,Goto, A.,Iga, Y.,Ikohagi, T., 2003."Cavitation CFD in a centrifugal pump," Proceedings of International Symposium on Cavitation. Osaka, Japan, pp. 1-7.

[24] Caridad, J., Asuaje, M.,Kenyery, F.,Tremante, A.,Aguillon, O., 2008. Characterization of a Centrifugal Pump Impeller under Two-Phase Flow Conditions. Journal of Petroleum Science and Engineering 63,p. 18.

[25] Gonzalez, J., Fernandez, J., Blanco, E., Santolaria C., 2002.Numerical Simulation of the Dynamic Effects Due to Impeller-Volute Interaction in a Centrifugal Pump.Journal of Fluids Engineering 124,p. 348.

[26] Wang, H., Tsukamoto, H., 2001. Fundamental Analysis on Rotor-Stator Interaction in a Diffuser Pump by Vortex Method.Journal of Fluids Engineering 123, p. 737.

[27] Salvadori, S., Gatta, S., Adami, P.,Bertolazzi, L., "Development of a CFD procedure for the axial thrust evaluation in multistage centrifugal pumps."

[28] Gatta, S.,Salvadori, S., Adami, P.,Bertolazzi, L., 2006."CFD study for assessment of axial thrust balance in centrifugal multistage pumps," International Conferenceon Fluid Flow Technologies.

[29] Wilson K., Addie G., Clift R., 1992. Slurry Transport Using Centrifugal Pumps.Elsevier Science Publishers Ltd., Essex, England.

[30] Pagalthivarthi, K., Gupta, P.,Tyagi, V., Ravi, M., 2011.CFD Predictions of Dense Slurry Flow in Centrifugal Pump Casings.International Journal of Aerospace and Mechanical Engineering 5 (4).

[31] Yu, S., Ng, B., Chan, W., Chua, L., 2000. The Flow Patterns within the Impeller Passages of a Centrifugal Blood Pump Model. Medical Engineering & Physics 22, p. 381.

[32] Williams, A., "Pumps as turbines for low cost micro hydropower."

[33] Natanasabapathi S.,Kshirsagar, J., "Pump as turbine - an experience with CFX-5.6."

[34] Fernandez, J., Barrio, R., Blanco, E.,Parrondo, J., Marcos, A., 2010. Numerical Investigation of a Centrifugal Pump Running in Reverse Mode. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy 224, p. 373.

[35] Barrio, R., Fernandez, J., Blanco, E., Parrondo, J., Marcos, A., 2011. Performance Characteristics and Internal Flow Patterns in a Reverse-Running Pump–Turbine. Proceedings of IMechE Vol. 226 Part C: J. Mechanical Engineering Science, p. 695.

[36] Liu, S.,Michihiro, N., Yoshida, K., 2001. Impeller Geometry Suitable for Mini Turbo-Pump.Journal of Fluids Engineering 123,p. 500.

[37] Tsui, Y., Lu, S., 2008.Evaluation of Performance of a ValvelessMicropump by CFD and Lumped System Analyses. Sensors and Actuators 148, p. 138.

[38] Shah, S., M Tech thesis. CFD Analysis of Centrifugal Pump. Institute of Technology, Nirma University, May 2010.