Available online at www.sciencedirect.com
SciVerse ScienceDirect
Procedia Engineering 51 (2013) 797 – 802
Procedia Engineering
www.elsevier.com/locate/procedia

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

Numerical optimization of guide vanes and reducer in pump running in turbine mode

V A Patel[a], S V Jain[b], K H Motwani[c], R N Patel[b]

[a]Essar Steel India Ltd., Surat-394270, India
[b]Institute of Technology, Nirma University, Ahmedabad-382481, India
[c]Faculty of Technology, Marwadi Education Foundation Group of Institutions, Rajkot-360003, India

Abstract

The main limitation of centrifugal pump running in turbine mode is lower efficiency at part load operating conditions as compared to conventional hydro turbines. To enhance the utilization of pump as turbine (PAT) in mini/micro hydropower plants, it is very important to improve its performance with some low cost modifications. The part load efficiency of PAT can be improved by guiding the flow at part load by installing the movable guide vanes. However, to avoid the higher cost of movable guide vanes, it was proposed to install fixed guide vanes. The existing volute casing of centrifugal pump was not having sufficient space for provision of guide vane mechanism. Hence, to create an additional space the existing impeller of 250 mm dia. was replaced by 200 mm dia. impeller which has facilitated an additional space for installation of guide vanes. To determine the optimum position of fixed guide vanes, CFD analysis of casing was carried out with NACA-4418 profile by varying the guide vane angles between 45° to 80°. The numerical simulations were carried out using Reynolds averaged Navier-Stokes (RANS) equations with standard k-ε turbulence model under steady state conditions. In addition, it was required to place the reducer between the service pump and PAT as the pipe dia. at service pump outlet was 5” and at PAT inlet was 3”. To optimize the shape and location of reducer between service pump and PAT, CFD analysis was performed by considering two different reducers at different locations. PAT with 200 mm dia. impeller and 8 numbers of fixed guide vanes provided at 75° angle led to improved performance of PAT. Also the long reducer provided at the inlet of PAT was subjected to less head drop in the piping system.

Keywords: Guidevane; numerical simulation; optimization; pump as turbine.

1. Introduction

Energy plays an important role in almost all areas of human and commercial activities and it is very important input for those countries that are developing from economic point of view. Worldwide, around 67% electricity is generated from fossil fuels [1]; however, high prices, fast depletion rate and environmental implications of fossil fuels create problems in electricity generation through conventional energy sources. Electricity generation through renewable energy sources is appropriate solution for these problems [2]. In addition to this, economic development through renewable energy industry and sustainable energy sector create more employment, which leads to social development of the nation [3].

One of the most promising non conventional energy that can be used for generating power in this situation is hydro technology. Hydropower is a non conventional, non-polluting and environmentally being source of energy [4]. The Ministry of Non-Conventional Energy Resources, Government of India identified several sites in North India particularly in the Himalayan Range. The streams flowing from a height of 30 to 50 meters with relatively low discharge were

* Corresponding author. Tel.: +91-9998623087; fax: +91-2717-241917.
E-mail address: sanjay.jain@nirmauni.ac.in

doi:10.1016/j.proeng.2013.01.114

considered suitable to generate power in the range of 15 to 50 kW [5], which falls in the category of micro hydropower (less than 100 kW) [6]. In developing countries, small and micro hydropower plants are very effective source for electricity generation. The energy pay-back time (EPBT) and greenhouse gas (GHG) emissions for SHP generation system are less than other conventional electricity generation system [7]. So, encouragement of small hydropower schemes can solve the problem of energy crises of the country.

The problems associated with small/mini/micro hydropower development are technical and economical as compared to conventional power plants. The design of a single machine and its manufacturing to achieve maximum efficiency for each small hydro plant site is costly affair. The electro-mechanical equipment of such plants constitutes about 60-70% of the total project cost. Hence minimising the cost of this equipment can reduce cost of the plant [8]. Centrifugal pump working as turbine is a good alternative for power generation through mini and micro hydropower schemes. Pumps offer many advantages over custom-made turbines that make their use an economical solution in this sector. The market for pumps is large and being a standard product it is always cheap and readily available. Also, operational and maintenance issues in pumps are relatively simpler as compared to turbines [5].

Hydroelectric power plants are often subjected to run at off-design operation in order to satisfy the fluctuating load demands which would naturally require higher performances at part loads.

The main limitation of PAT is lower efficiency at part load operating conditions as compared to conventional hydro turbines. To enhance the utilization of PAT in mini/micro hydropower plants, it is utmost important to improve its performance with some low cost modifications.

Many researchers have attempted different modifications to improve the performance of PAT by carrying out experimental and numerical investigations. Derakhshan et al. [9] redesigned the shapes of the blades using a gradient based optimization method involving incomplete sensitivities for radial turbo machinery to obtain higher efficiency. Singh [10] demonstrated various possibilities of modifying the pump geometry to improve the performance in turbine mode viz. by rounding of inlet edges of impeller, by modifying the inlet casing rings, by enlarging the suction eye, by removing the casing eye rib. Singh and Nestmann [11] studied the effects of impeller rounding on 9 PATs covering a specific speed range of 20 to 94.4 rpm. Suarda et al. [12] experimentally determined the performance of a small centrifugal volute pump running in reverse mode after grinding the inlet ends of the impeller tips to a bullet-nose shape. Sheng et al. [13] applied splitter blade technique to improve the PAT performance which is one of the techniques used in flow field optimization and performance enhancement of rotating machinery. Williams and Rodrigues [14] discussed the effects of enlargement of the suction eye of the pump by removing the material.

The part load efficiency of PAT can be improved by guiding the flow at part load operating conditions by installing the guide vanes. In the present study, two different attempts were made to improve the performance of PAT. In the first method, the guide vane positions were optimized by carrying out numerical simulation of casing by putting the guide vane at different angles. In the second approach, the shape and location of reducer, between service pump and PAT, was optimized by performing CFD analysis by considering two different reducers at different locations.

2. Numerical simulation of PAT [16]

One of the most reliable tools used by many researchers to understand flow behavior in the hydro-mechanical equipment is CFD. It helps to predict fluid flow by means of mathematical modeling, software tools and numerical methods. These numerical methods help to investigate various parameters such as flow pattern and hydraulic losses which cannot be easily measured by experiment also. In this paper, the effects of different modifications on the performance of PAT are studied using CFD as a numerical simulation tool. The end suction, single stage centrifugal pump (head: 15 m, discharge: 1500 LPM) was used as a turbine, by running it in reverse direction, for the experimental and numerical investigations. The view of the centrifugal pump and two-dimensional drawing of casing are shown in Fig. 1.

The existing volute casing of centrifugal pump was not having sufficient space for provision of guide vane mechanism. Hence, to create an additional space the existing impeller of 250 mm dia. was replaced by 200 mm dia. impeller which has facilitated an additional space for installation of guide vanes. Considering the geometry of the casing, NACA 4418 was selected for the analysis as per the guidelines given in [15]. Initially, the numerical simulations of casing were carried out by considering the space for 250 mm and 200 mm dia. impellers. The two-dimensional (2D) geometry of the computational domain was created in GAMBIT, which is a pre-processor of FLUENT, using bottom-up approach. The numerical models of casing with space for different impellers and the guide vane profile are shown in Fig. 2.

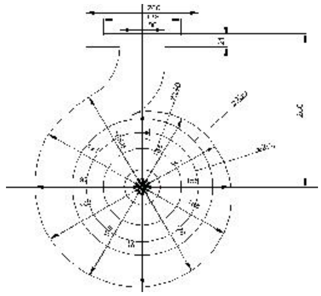
(a)  (b) 

Fig. 1. (a) Centrifugal pump (left) and (b) 2-D drawing of casing (right).

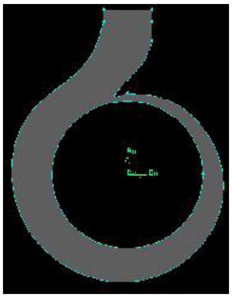
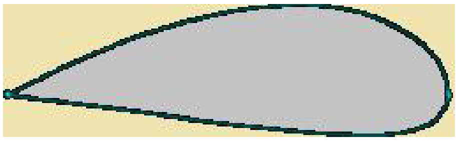
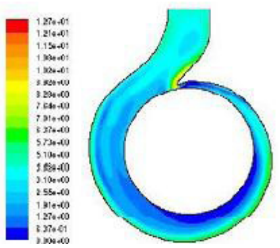
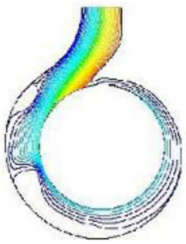
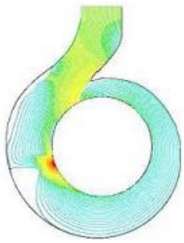
(a)  (b)  (c) 

Fig. 2. Numerical models of casing with space for impellers of (a) 250 mm dia. (b) 200 mm dia. and (c) 2D view of guide vane (NACA 4418) profile.

The flow domain was discretized into around 2.2 lacs quadrilateral elements of 0.5 mm size. Mass and momentum conservation equations were discretized using first order upwind scheme and solved in 2D double precision segregated solver. To consider the turbulence effects in the casing standard k-ε turbulence model was used and SIMPLE algorithm was applied as pressure-velocity coupling. As boundary conditions, mass flow inlet and outflow were specified at casing inlet and outlet respectively. In both the cases the inlet velocity was kept as 3.978 m/s. At design condition, the steady state simulations were performed using two equations Reynolds averaged Navier-Stokes (RANS) models by considering the space for 250 mm and 200 mm dia. impellers in the casing. The velocity contours and path lines in the casing with space for both the impellers are shown in Fig. 3. It was found that, the drop in kinetic energy in the casing is less with smaller impeller. Also, the smaller impeller facilitates an additional space for the provision of guide vanes.

(a) (b) (c) (d)

Fig. 3. Velocity contours and path lines in casing with 250 mm (a & b) 200 mm (c & d) dia. impellers.

By considering 200 mm impeller, depending upon availability of space in the casing, 8 numbers of guide vanes were proposed. Conventional hydro turbines are provided with movable guide vanes; however, in case of PAT the movable guide vanes may lead to higher cost. Hence, from cost saving considerations, it was proposed to install fixed guide vanes. In case of fixed guide vanes, it was very crucial to optimize the angle of guide vanes; otherwise, it may lead to higher hydraulic losses at part load conditions. Thus, to decide the optimum position of fixed guide vanes, CFD analysis of casing was carried out by considering different guide vane angles viz. 45, 60, 75, 80 degrees. The total numbers of 2-D quadrilateral elements for the whole flow domain were around 2.7 lacs of 0.5 mm size. Size function was attached on the vanes to capture the higher gradients. The mesh for the casing with guide vanes at different angles is shown in Fig. 4.

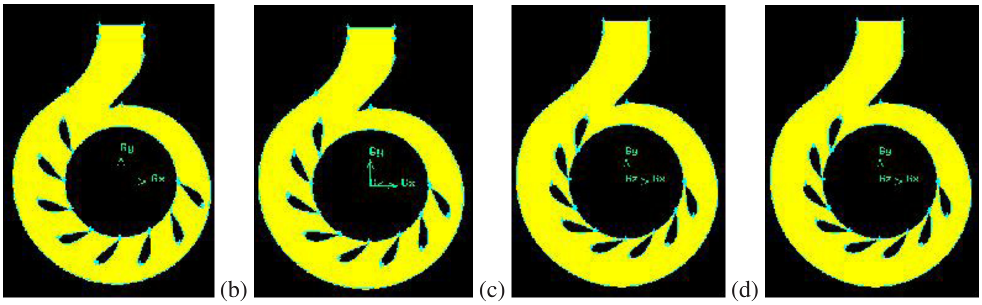


Fig. 4. Mesh for casing with guide vanes at (a) 45° (b) 60° (c) 75° (d) 80°.

The velocity contours in the casing with guide vanes at different positions are shown in Fig. 5. The boundary conditions and simulation parameters were kept same as in the case of simulations without guide vanes. It was observed that, as angle of guide vane increases percentage velocity drop in the casing decreases up to guide vane angle of 75°, *i.e. the loss in kinetic energy in the casing decreases which may facilitate availability of more energy for the runner*, and then increases. Hence, it was proposed to put guide vanes at an angle of 75° in the casing of PAT, *which facilitate tangential entry of water in the impeller*, for better performance. The variation of decrease in velocity in the casing with increase in guide vane angle is shown in Fig. 6.

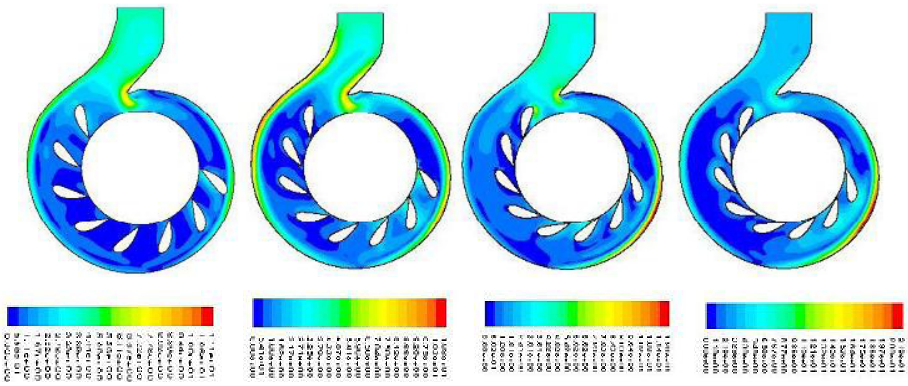


Fig. 5. Velocity contours in casing with guide vanes at (a) 45° (b) 60° (c) 75° (d) 80°.

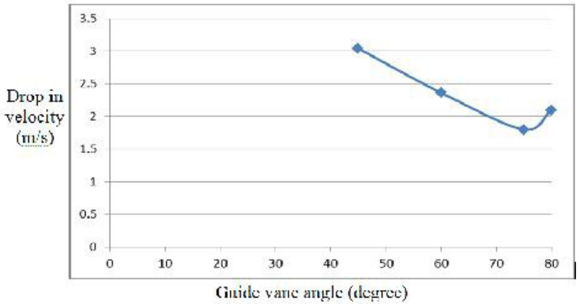


Fig. 6. Percentage drop in velocity with increase in guide vane angle.

3. Optimization of shape, size and location of reducer in the piping system [17]

The schematic diagram of experimental test rig developed at Institute of Technology, Nirma University is shown in Fig. 7. In the test rig, the pipe dia. at service pump outlet was 5” and at PAT inlet was 3”. Hence, it was required to place the reducer (of 5" X 3" dia. size) between the service pump and PAT. The head losses due to reducer depend on its length and the location between the service pump and PAT. To minimize the head losses in the reducer and to study the fluid behavior in the piping system, CFD analysis of two reducers was performed at two different locations. The first reducer was having length of 2”, *named as short reducer*, was placed between the service pump and PAT; and the second reducer was having 18” length, *named as long reducer,* was placed at the inlet of PAT as shown in Fig. 8.

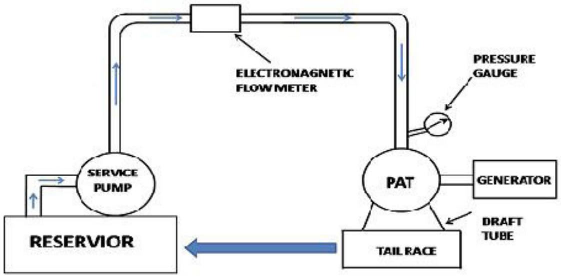


Fig. 7. Schematic diagram of experimental set up.

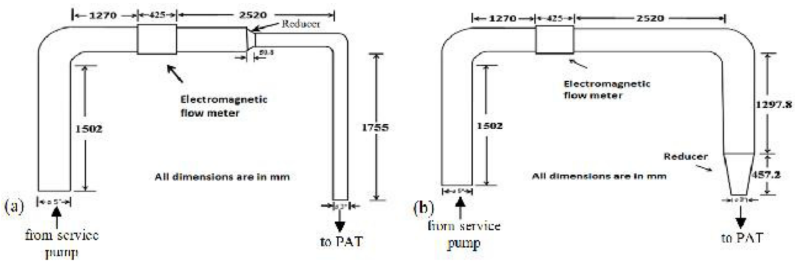


Fig. 8. Layout of piping system (a) short reducer between service pump and PAT and (b) long reducer at the inlet of PAT.

The three-dimensional computational domain was prepared in GAMBIT and was discretized into around 4.3 lacs quadrilateral elements of 0.5 mm size. As boundary conditions, velocity inlet and pressure outlet were specified at pipe inlet and outlet respectively; the values of which were achieved from the results of previously performed experiments. Other simulation parameters were kept same as described in section 2 above. Computational domain with boundary conditions and velocity vectors near the reducer are shown in Fig. 9; and the static pressure contours for both the cases are shown in Fig. 10. It was found that, by using the long reducer (at PAT inlet) in place of short reducer (between service pump and PAT), the loss in static and dynamic pressures in the piping system were reduced by 55.55% and 72.73% respectively. It has advocated that, the head losses in case of long reducer were less than that with short reducer. Hence, it was recommended to install the piping system with long reducer as shown in Fig. 8 (b).

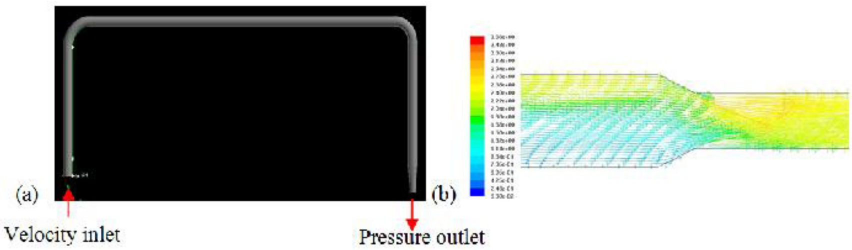
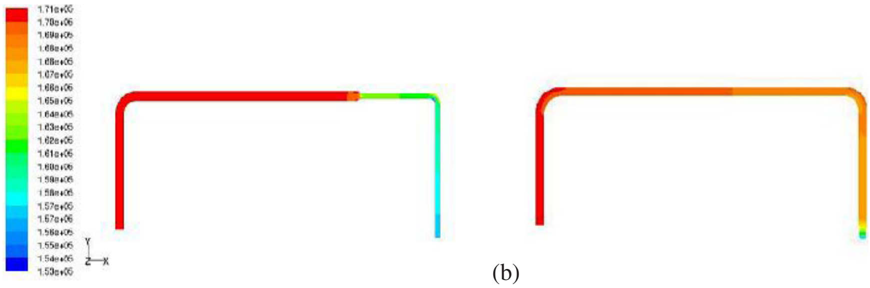


Fig. 9. (a) Computational domain and boundary conditions and (b) velocity vectors near the reducer.



(a) (b)

Fig. 10. Variation of static pressure in piping system with (a) short reducer between service pump and PAT and (b) long reducer at the inlet of PAT.

4. Conclusions

One of the reasons for poor part load efficiency of pump running in turbine mode is absence of flow regulating mechanism, which can be improved by some low cost modification like by providing the fixed guide vanes in the casing. To create the space for the installation of guide vanes in the casing, it was proposed to use 200 mm dia. impeller in place of 250 mm dia. impeller. To determine the optimum angle of fixed guide vanes, CFD analysis of casing was carried out with NACA-4418 guide vanes by varying the guide vane angle between 45° and 80°. At an angle of 75°, the loss in kinetic energy in the casing was found to be minimum, *as the fluid enters tangentially in the runner blades*, which has suggested availability of more energy for the power generation to the runner. Hence, it was recommended to use 200 mm dia. impeller along with 8 numbers of fixed guide vanes angle at an angle of 75° for the PAT. To decide the size and location of reducer in the piping system, CFD analysis of piping system was carried out by considering two diffusers i.e. short reducer between service pump and PAT and long reducer at PAT inlet. With long reducer, the loss in static and dynamic pressures in the piping system was found to be 55.55% and 72.73% less than that with the short reducer. Hence, use of long reducer was recommended at the inlet of PAT.

References

[1] http://www.powermin.nic.in/indian_electricity_scenario/introduction.htm.
[2] Nautiyal, H., Varun, Kumar, A., Yadav, S., 2011. Experimental Investigation of Centrifugal Pump Working as Turbine for Small Hydropower Systems. Energy Science and Technology 1 (1), Canada, p. 79.
[3] Akella, A., Saini, R., Sharma, M., 2009. Social, Economical and Environmental Impacts of Renewable Energy Systems. Renewable Energy 34 (2), p. 390.
[4] Jain, S., Singal, S., 2009. "Design of water conductor system for ultra low head hydropower project – a case study," Proceedings of the International Conference on Advances in Mechanical Engineering, Gujarat, India.
[5] Rawal, S., Kshirsagar, J., 2007. "Numerical simulation on a pump operating in a turbine mode," Proceedings of 23rd International Pump Users Symposium, pp. 21- 27.
[6] Small and Mini Hydel Projects, Sixteenth Report, Standing Committee on Energy, Ministry of New and Renewable Energy, 2010-11.
[7] Varun, Bhat, I., Prakash, R., 2008. Life Cycle Analysis of Run-of River Small Hydro Power Plants in India. The Open Renewable Energy Journal 1, p. 11.
[8] Prasad, V., Shukla, S., Joshi, S., 2006. Performance Characteristics of Pump as Turbine. Indian Pumps.
[9] Derakhshan, S., Mohamadi, B., Nourbakhsh, A., 2009. Efficiency Improvement of Centrifugal Reverse Pumps. Journal of Fluids Engineering 131, p. 1.
[10] Singh, P., Ph.D. thesis: Optimization of the Internal Hydraulic and of System Design in Pumps as Turbines with Field Implementation and Evaluation," University of Karlsruhe, Germany, June 2005.
[11] Singh, P., Nestmann, F., 2011. Internal Hydraulic Analysis of Impeller Rounding in Centrifugal Pumps as Turbines. Experimental Thermal and Fluid Science 35, p. 121.
[12] Suarda, M., Suarnadwipa, N., Adnyana W., Experimental Work on Modification of Impeller Tips of a Centrifugal Pump as a Turbine. Udayana University Denpasar, Bali, Indonesia.
[13] Sheng, Y., Yu, K., Hui, F., Ling, X., 2012. Numerical Research on Effects of Splitter Blades to the Influence of Pump as Turbine. International Journal of Rotating Machinery 2012, p. 1.
[14] Williams, A., Rodrigues, A., 2008. The Performance of Centrifugal Pumps as Turbines and Influence of Pump Geometry.
[15] Riegels F., 1961. Aerofoil sections. Butterworth and Co. Publishers Ltd., London.
[16] Patel, V., M Tech thesis, Performance Enhancement of Existing Pump as Turbine. Mechanical Engineering Department, Institute of Technology, Nirma University, May 2011.
[17] Motwani, K., M Tech thesis, Experimental and Numerical Investigations on Pump as Turbine for Micro Hydropower Plants. Mechanical Engineering Department, Institute of Technology, Nirma University, May 2012.