

CFD Analysis of Pump as Turbine for Micro-Hydro Schemes

E.Baburaj¹, R.Sivaprakasam², C.Manikandan³, K.Sudha⁴

Assistant Professor, Department of Mechanical Engineering, Karpagam College of Engineering Coimbatore,
Tamil Nadu, India^{1,2,3&4}.

ABSTRACT: A pump can be used as turbine and has good application in micro-hydropower schemes. Pump-as-turbine (PAT) is one of the best alternatives for fulfilling the energy demands and providing the electricity in remote and rural areas. Economic development through renewable energy and sustainable energy sector generate additional employment, which leads to social improvement of the nation. One of the reasons for the micro hydro left undeveloped is because of economic constraints. Although the running cost for micro hydro scale are very low, the initial capital cost are high especially the turbines cost. Using pump-as-turbine (PAT) is an attractive and significant alternative. In this paper, theoretical analysis of the relationship between pump and PAT performance was performed. A hypothetical method of predicting PAT performance is developed using theoretical analysis and empirical correlation. Subsequently, computational fluid dynamics (CFD) was adopted in the direct and reverse modes performance prediction of a centrifugal pump.

KEYWORDS: Centrifugal Pump, Pump as turbine, Computational Fluid Dynamics.

I. INTRODUCTION

Energy plays a significant role in almost all areas of human and commercial activities, and it is particularly main input for those countries that are improving from financial point of view. The generation of electrical energy is essential to raise the economy's infrastructure of a country. Generation of power through the non-renewable energy source is quite familiar. Several thermal power plants are working all over the world for electrical energy generation but the rapid depleting environment and growing prices of petroleum products (coal, oil, gas, etc.) are the almost important complication in satisfying the power demands from these sources [1]. Electrical energy supply generated in world for about 20,053 TWh of installed capacity and that to hydro power contributes to around 16% in many countries it is the main source of power generation [2].

Economic growth through renewable energy creates more employment that leads to social improvement of the country. Almost promising non conventional energy for generating power in this condition is hydro machinery. The Ministry of Non-Conventional Energy Resources, Government of India recognized numerous sites in North India, generally in the Himalayan Range for the development of micro hydro power plants ranging from 15 to 50 kW [3]. In the present paper, a simple theoretical method to predict the BEP of PAT using theoretical analysis and empirical correlation on the basis of its pump performance is developed. The PAT performance is very complex and it is difficult to find a relation to cover all pumps performance in reverse mode. One idea is using full computational fluid dynamics to simulate PAT performance. Mainly impeller and volute has been simulated in direct and reverse mode operation.

II. PAT FOR HYDRO POWER GENERATION

When the centrifugal pump works in reverse mode it simply reduce the equipment cost and can be used as an alternative to conventional turbine [1]. Pump manufactures do not generally provide the characteristic curves of their pumps in reverse action. Consequently, establishing a correlation enabling the passage from the "pump" characteristics to the "turbine" characteristics is the most important dispute in using a pump as a turbine. The hydraulic performance of a pump when rotates as a turbine will be changed. In general a pump will operate in turbine mode with higher head and discharge in the same rotational speed. Several researchers have offered some theoretical and experimental relations for predicting the PAT characteristics in the best efficiency point (BEP) [4].

Micro hydro power can be one of the most important alternative to isolated rural communities the advantages of electrification and the associated progress, as well as to improve the quality of life. It is one of the most commercial hydro power energy technologies for rural electrification in less developed countries [5].

The MNRE has renowned more than 6000 streams in north-east and north regions of India, that are not suitable for installation of large hydro power plants but are favourable for electricity generation in the range of 5–100kW. Mini, micro and small hydro power schemes can be installed on such water streams [6].

III. LITERATURE REVIEW

Williams [7] investigated different forms of prediction methods for the effective use of PAT. The experimental study were conducted for more than 30 pumps of different standards and suction methods. The result of the particular study clearly provide that no method will be suitable for the prediction of generalised PAT. Yang et al., [4] investigated the insufficient understanding of the correlation between pump and pump as turbine (PAT) performance is a major problem encountered in the PAT selection.

Singh and Nestmann [8] developed an efficient model to study the rounding of the sharp edges at the impeller edge on a combination of radial flow and mixed flow pumps as turbines using experimental parameter. Proper rounding of impeller selection provide optimistic answer to the overall efficiency in all operating condition.

Derakhshan et al., [9] derived a latest technique to forecast the BEP of a PAT based on pump's hydraulic specifications. Derakhshan and Nourbakhsh, [10] calculated the 'area ratio' method for best efficiency point in theoretical investigation. Each and every part of pump were estimated by geometric parameter and through performance curves.

Rodrigues et al., [11] worked on computational fluid dynamics to know about the flow conditions and hydraulic performance of PAT. The numerical study is focused on the casing of pump, inlet and outlet zone, impeller diameter and draft tube. CFD provides valuable result when compared with experimental result with some error ranging upto 10% for different part load conditions.

Natanasabapathi and Kshirsagar [12] investigated PAT using CFD approach. It was concluded that the impractical consequences can be obtained from amorphous gridding across the interface between stationary and revolving structure. Thus, structured gridding close to the interface is a resolution to minimize such errors in the results.

Rawal and Kshirsagar [6] carried out numerical analysis on axial type pump running in turbine mode. Resemblance was established between the numerical and experimental outcome for best efficiency point. The numerical system was quite supportive in study of different parameters like losses in the impeller, casing, draft tube and flow pattern which could not be calculated through experiments.

Derakhshan and Nourbakhsh [10] A centrifugal pump ($NS = 23.5$ (m, m³/s)) was simulated using CFD in direct and reverse modes. To verify numerical results, simulated pump was tested as a turbine in the test rig. Characteristics of best efficiency point predicted by numerical methods were not in good coincidence with experimental data. Future works on CFD application can be improved, since the application of CFD fails in the turbine boundary as proved with this study.

Nautiyal et al., [13] study on the application of numerical methods and its restrictions. Investigation reported that CFD study was an efficient design tool for predicting the performance of centrifugal pumps in turbine mode and to identify the losses in turbo components.

Barrio et al., [14] worked out a numerical analysis on the uneven flow in commercial centrifugal pump working in forward and reverse direction mode with the help of CFD tool. Outcome of mathematical simulation were in good agreement with the experimental results. The study clearly explains that in the reverse mode, the flow corresponding to the impeller geometry at normal conditions while developing re-circulating fluid regions at high and low flow rates. Fecarotta et al., [15] studied the least difficulty of the CFD computation mesh to perform faster and consistent simulations. The investigation described the CFD study as reliable tool to better understand the interaction between the hydro mechanical components and the flow performance in spite of complex computation. CFD calculations were carried out to forecast the turbine performance under various flow conditions and the characteristic curves for pump and turbine modes were obtained.

Agarwal [16] noted computational Fluid Dynamics (CFD) is an efficient design tool for predicting the performance of centrifugal pumps operating in turbine mode. It has been established that the results from numerical and experimental investigation do not match exactly in case of PAT. It is recommended that the variation can be minimized through development in computational study by using finer mesh, numerical methods and turbulences models. CFD study is helpful in identifying the losses in hydraulic machine components.

IV. PUMP DESIGN PARAMETERS

The focus of the investigation is a single stage centrifugal pump with a rotational speed of 2880 rpm. Table 1 lists the main geometric parameters of the designed pump.

Table1. Main geometric of the designed pump

H	N _s	D _s	D _H	D ₁	D ₂	β ₁	β ₂	b ₁	b ₂	Z
45	23.9	10	40	81	203	19.8°	25°	18	9	6

H head, m

β₁ blade inlet angle (°)

D_s impeller shaft diameter, mm

D_H impeller hub diameter, mm

D₁ impeller inlet diameter, mm

D₂ impeller outlet diameter, mm

Z number of vanes

N_{sp} specific Speed

β₂ blade outlet angle (°)

b₁ inlet blade height, mm

b₂ outlet blade height, mm

V. SIMULATION

The numerical analysis of radial pump with specific speed of 23.9 is conducted in software ANSYS-CFX. ANSYS-CFX is a commercial 3D Navier-stokes CFD code that utilises a finite-element-based finite-volume method to discretise the transport equations. It is a completely embedded solver, thus it creates no time step restriction and is measured easy to recognize. It is also a united solver meaning that the momentum and continuity equations are solved. This approach reduces the number of steps required to obtain convergence and no pressure improvement term is required to retain mass alteration, leading to a more healthy and correct solver.

A. Meshing

The meshing is done with aid of mesh for CFD in Ansys workbench. It is unstructured mesh consists of quadrilateral and triangular elements. Number of elements for impeller is

1, 77,000 and for volute is 1, 30,000.

B. Pre-processing

Inlet boundary condition commonly used for pump analysis is total pressure. In our case, we are using total pressure is as environmental pressure [1 bar]. Outlet boundary conditions commonly used are static pressure or mass flow rate. In our case, we are using mass flow outlet as per software requirement. This mass flow rate is continuously varied to get corresponding head at outlet of pump to draw performance curve. Because of memory consideration impeller and volute are individually analyzed. Total pressure at impeller outlet is given to the volute as inlet condition.

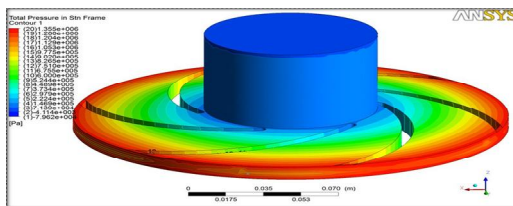


Fig.1. Impeller-total pressure at 0 Kg/s

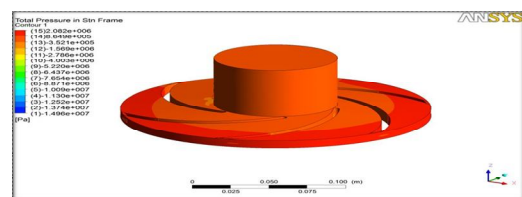


Fig.2. Impeller-total pressure at 20.83

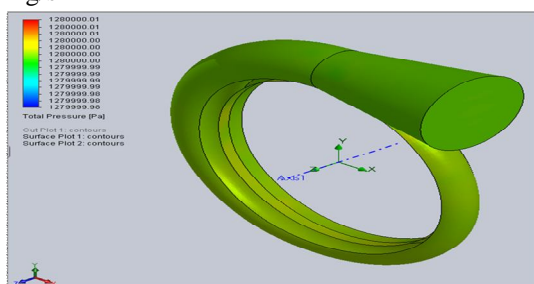


Fig.3. Volute- total pressure at 0 Kg/s

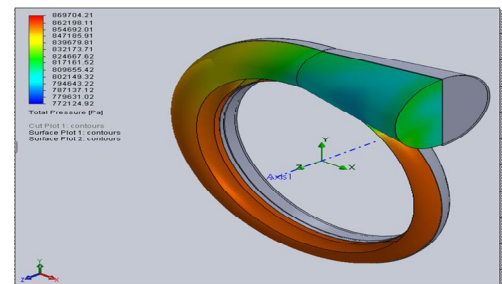


Fig.4. Volute-total pressure at 20.83

VI. RESULT AND DISCUSSION

Impeller results are given below. It shows total pressure outlet of impeller for different mass flow rate outlet.

Table 2. Q & H impeller

Q (Kg/s)	35	25	20.83	15.83	10.83	5.83	0
H (m)	60	75	86	90	100	108	128

These impeller outlets are given to volute as inlet condition.

Table 3. Q & H Volute

Q (Kg/s)	35	25	20.83	15.83	10.83	5.83	0
H (m)	46.70	68.19	81.27	87.26	98.71	107.62	128

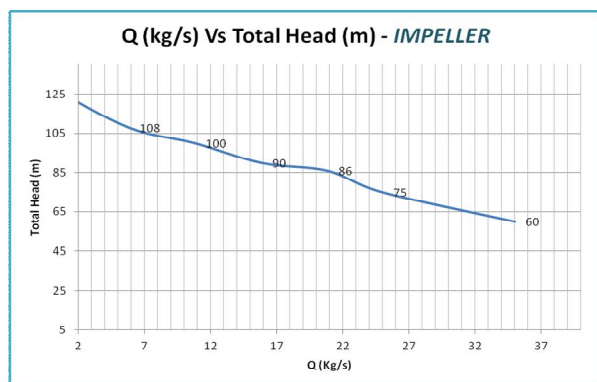


Fig.5. Q(Kg/s) vs H (m) Impeller

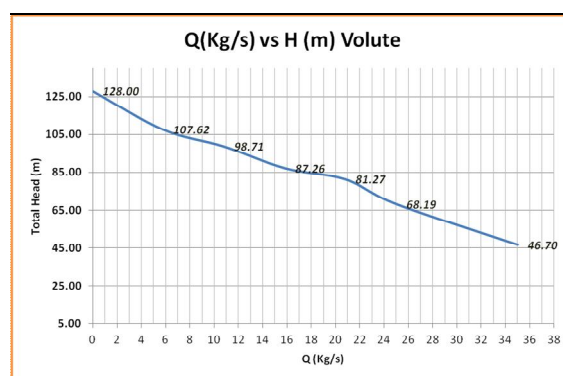


Fig.6. Q(Kg/s) vs H (m) Volute

The following relationship are used to find the mass flow rate, Specific speed of pump, Specific speed of turbine, Efficiency of turbine.

Mass flow rate, $M = Q * \rho$

Specific speed of pump, $N_{sp} = (N Q^{0.5}) / H^{0.75}$

Specific speed of turbine, $N_{st} = (N_{sp} - 1.73) / 1.125$

Efficiency of turbine, $\eta_{turbine} = (N_{st} / N_{sp})^2 * 100$

Table 4. Q, N_{sp} , N_{st} , $\eta_{turbine}$

Q (Kg/s)	35	25	20.83	15.83	10.83	5.83	0
N_{sp}	31	26.2	23.9	20.85	17.25	12.65	0
N_{st}	26	21.75	19.7	17	13.8	9.7	0
turbine	70.4	68.9	67.9	66.4	63.9	58.7	0

VII. CONCLUSIONS

The current study will provide complete details for selection of centrifugal pump as turbine for micro-hydropower plants. From the table 4. that the efficiency of turbine is attained upto 70% when the centrifugal pump operated in reverse mode. The future work for further development of PAT performance will be on low cost modifications in the impeller, fluid structure interaction, casing quality, impeller eye clearance, bearing quality, selection of material for shaft and impeller. Characteristics of best efficiency point can be improved by using various modification recommended by earlier researchers will provide enhanced understanding of PAT.

REFERENCES

- [1]H. Nautiyal and A. Kumar, "Reverse running pumps analytical, experimental and computational study: A review," *Renewable and Sustainable Energy Reviews*, vol. 14, pp. 2059-2067, 2010.
- [2]S. V. Jain and R. N. Patel, "Investigations on pump running in turbine mode: A review of the state-of-the-art," *Renewable and Sustainable Energy Reviews*, vol. 30, pp. 841-868, 2014.
- [3]V. Patel, S. Jain, K. Motwani, and R. Patel, "Numerical Optimization of Guide Vanes and Reducer in Pump Running in Turbine Mode," *Procedia Engineering*, vol. 51, pp. 797-802, 2013.
- [4]S.-S. Yang, S. Derakhshan, and F.-Y. Kong, "Theoretical, numerical and experimental prediction of pump as turbine performance," *Renewable Energy*, vol. 48, pp. 507-513, 2012.
- [5]O. Paish, "Micro-hydro power: status and prospects," *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, vol. 216, pp. 31-40, 2002.
- [6]S. Rawal and J. Kshirsagar, "Numerical simulation on a pump operating in a turbine mode," in *Proceedings of the 23rd International Pump Users Symposium*, 2007, pp. 21-27.
- [7]A. Williams, "The turbine performance of centrifugal pumps: a comparison of prediction methods," *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, vol. 208, pp. 59-66, 1994.
- [8]P. Singh and F. Nestmann, "Internal hydraulic analysis of impeller rounding in centrifugal pumps as turbines," *Experimental Thermal and Fluid Science*, vol. 35, pp. 121-134, 2011.
- [9]S. Derakhshan, B. Mohammadi, and A. Nourbakhsh, "Efficiency improvement of centrifugal reverse pumps," *Journal of fluids engineering*, vol. 131, 2009.
- [10]S. Derakhshan and A. Nourbakhsh, "Theoretical, numerical and experimental investigation of centrifugal pumps in reverse operation," *Experimental Thermal and Fluid Science*, vol. 32, pp. 1620-1627, 2008.
- [11]A. Rodrigues, P. Singh, A. Williams, F. Nestmann, and E. Lai, "Hydraulic analysis of a pump as a turbine with CFD and experimental data," in *IMEchE seminar computational fluid dynamics for fluid machinery*, 2003.
- [12]S. Natanasabapathi and J. Kshirsagar, "Pump As Turbine-An Experience With CFX-5.6," *Corporate research and engineering division. Pune, India, Kirloskar Bros. Ltd.*, 2004.
- [13]H. Nautiyal, V. Kumar, and A. Thakur, "CFD analysis on pumps working as turbines," *Hydro Nepal: Journal of Water, Energy and Environment*, vol. 6, pp. 35-37, 2010.
- [14]R. Barrio, J. Fernández, J. Parrondo, and E. Blanco, "Performance prediction of a centrifugal pump working in direct and reverse mode using Computational Fluid Dynamics," in *International Conference on Renewable Energies and Power Quality, Granada, Spain*, 2010.
- [15]O. Fecarotta, A. Carravetta, and H. Ramos, "CFD and comparisons for a pump as turbine: mesh reliability and performance concerns," *Int J Energy Environ*, vol. 2, pp. 39-48, 2011.
- [16]T. Agarwal, "Review of Pump as Turbine (PAT) for Micro-Hydropower," *International Journal of Emerging Technology and Advanced engineering*, vol. 2, pp. 163-168, 2012.