Investigation of the performance of pump as turbine by back cavity filling

A. P. Bahod, A. M. Belgi, K. V. Magdum, O. S. Sawant, M. B. Patil, S. G. Waghamare

Annasaheb Dange Collage of Engineering and Technology, Ashta, Maharastra, India.

Abstract: **Pump as turbine (PAT) is the best alternatives for the conventional turbine in micro hydro power plants. The objective of the present paper is to analyze the flow physics and estimation of the efficiency of PAT when the shape and size of the back cavity is changed. The design optimization of back cavity is carried out by numerical method. The efficiency of PAT is increased after filling the back cavity with a solid component. The efficiency increases mainly due to reduction of disk friction losses.**

I. INTRODUCTION

A. Background:

Hydropower is one of the renewable, green and reliable energy sources. With the technology of over 100 years, hydropower is considered as the most efficient technology to provide in large or small scale. Electricity requirement in many countries is mainly fulfilled by large hydropower plants. While, micro hydropower plants (MHP) are useful for the electrification of rural community, remote locations etc., and supplies power in the range of 5-100 KW. The ministry of renewable energy resources has searched near about 6000 streams in the northern and northern east India. These streams are small as compared to requirement of large hydropower. Hence, these streams can be used for micro hydro power plant. The use of conventional turbine for micro hydro power plant is not feasible. The main limitation of conventional turbine is its cost. The pump working in reverse mode as a turbine (PAT) is an alternative solution for the conventional turbines because of its low cost and availability in the market with wide range of head and discharge. The use of PAT dates back to 1920's without any formal studies of the turbine mode performance characteristics. Kittredge accidently discovered PAT using the testing of pump to arrest water hammer effect for large pump station [1]. A standard centrifugal pump when operated in reverse mode as a turbine to generate shaft work is called pump as turbine (PAT). In contrast to the conventional centrifugal pump in PAT fluid enters from delivery side of pump and flows through casing, impeller and finally exit through suction side of pump. PAT has advantages as it is simple in construction, easily available, low maintenance and installation cost and less complicated to operate in comparison to conventional turbines. Nowadays, PAT finds application in the energy recovery systems, water distribution and sewage systems, small pumped storage power plants, micro hydro power plants etc.

B. Internal flow physics of PAT:

PAT mainly consists of three zones. 1. Flow zone 2. Non-flow zone, and 3. Transition zone as presented in Fig. 1.

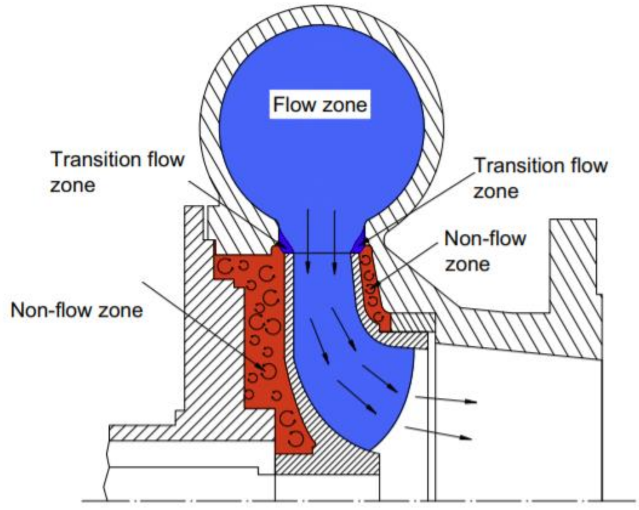

Figure 1 Flow control volumes of PAT [2]

Many researchers have studied various zones of PAT in order to improve the performance of PAT. Researchers have done many modifications in these zones. In 2018, Doshi [2] carried out experimental testing on back cavity filling. Doshi modified PAT considering non-flow zone especially back cavity (i.e. zone between outer shroud and casing). The novel method of filling back cavity by component with appropriate shape and size was proposed in order to improve the performance of PAT. However the actual flow pattern inside the PAT was not discovered. Also shape optimization of PAT is rarely discussed by the researchers.

C. Problem definition:

The main objective of this paper is to determine the performance of PAT after back cavity filling at best efficiency point (BEP).

II. METHAMETICAL MODEL

A. Theoretical models to analyse the performance of PAT:

In this section, theoretical models proposed by Doshi [2] to correlate the externally measured variables (head, speed, torque and discharge) with internal hydraulics (losses and net rotational momentum) is discussed.

1) Impeller torque or Euler head (HE):

Impeller torque or Euler head is the energy imparted from fluid onto PAT.

$$H_E = \frac{C_{u1}u_1 - C_{u2}u_2}{g} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots(3.1)$$

$$\Delta C_u r = C_{u1}r_1 - C_{u2}r_2$$

For constant diameter and steady speed PAT operation, rotational speed of runner is constant.

$$H_E = \left(\frac{\Delta(C_u u)}{g}\right)_{impeller} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots(3.2)$$

2) Net head (H):

Net head is the energy available between inlet and outlet section of PAT. Due to flow zone losses the impeller torque (head) or Euler head is always less than net head.

$$H = \left(\frac{\Delta(C_u u)}{g}\right)_{impeller} + h_{L,fz} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots..(3.3)$$

3) Shaft torque:

It is measured at end of the PAT shaft thereby it considers the loss in non-flow zone and the transition zone (neglecting mechanical losses). Therefore shaft torque can be obtained by subtracting the non- flow zone and transition zone losses from impeller torque.

Shaft torque = Impeller torque – head loss in the non-flow zone - head loss in the transition zone

$$\left(\frac{\Delta(C_u u)}{g}\right)_{shaft} = \left(\frac{\Delta(C_u u)}{g}\right)_{impeller} - h_{L,nfz} - h_{L,tz} \quad \ldots..(3.4)$$

Where, $\left(\frac{\Delta(C_u u)}{g}\right)_{shaft} = \frac{T\omega}{\rho Q g}$

Combining equation (3.3) and (3.4), net head on PAT can be obtained as

$$H = \left(\frac{\Delta(C_u u)}{g}\right)_{shaft} + h_{L,fz} + h_{L,nfz} + h_{L,tz} \quad \ldots\ldots\ldots\ldots...(3.5)$$

Total head loss $h_L = h_{L,fz} + h_{L,nfz} + h_{L,tz}$

$$H = \left(\frac{\Delta(C_u u)}{g}\right)_{shaft} + h_L$$

$$h_L = H - \left(\frac{\Delta(C_u u)}{g}\right)_{shaft} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots..(3.6)$$

B. Disk friction loss:

The power absorbed by disk friction is 2% of the total power [4]. At specific speed of 10 the disk friction power loss is about 50% of the total power, while at sp. speed 30, this fraction is about 5%. It is major contributor of the total losses for low to moderate specific speed pumps. Similarly pump disk friction losses also incur in PAT. Though, many researchers have studied the effect of non-flow zone on the pump performance, there is very less work available for pump operated in reverse mode. In this section, theory of disk friction losses and its evaluation by theoretical model is discussed.

1) Physics of disk friction losses:

Figure 2 shows the casing of diameter D filled with liquid; consist of a flat circular disk at the center, spaced at a distance B from either side of the casing. The disk is rotating with constant angular speed ω. When the disk is rotating, the liquid in the region bounded by the external surfaces of the disk and the end walls of the casing, moves outward due to application of the circumferential frictional forces and radial centrifugal force. This is replaced by liquid flowing towards the center along the end walls of the casing. Circulation takes place between both the sides of the disk surface and casing. The circulation mainly causes the disk friction losses, which depends on the clearance path between disk surface and casing. Shortest is the clearance path, lesser is the frictional loss [3].

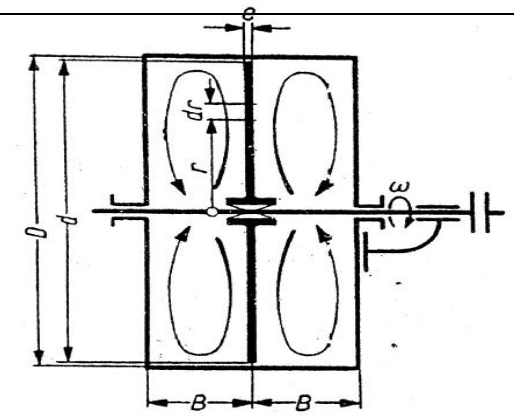

Figure 1 Rotating disk in cylinder containing liquid [3]

III. Neumerical Analysis

A. Solid Modelling of PAT:

The process of preparing or creating a computational model from a 3D model of an object is called as solid modeling. Solid modeling of an object is generally done to have geometrical correctness of the proposed object. From geometry view, it is very difficult to create geometry of PAT so with the help of CMM (Co-ordinate Measuring Machine) we measured each and every point of PAT and converted into the solid model with the help of solid works. Below fig shows the solid model of PAT.

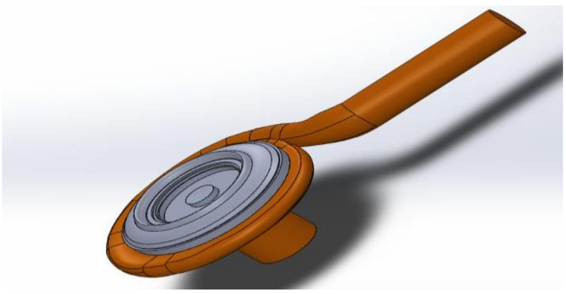

Figure 2 Solid model

B. Computational Model of PAT:

Conversion of solid model into a model ready for simulation and computation is called as computational modelling. A computational model can directly be used for analysis and simulation purpose. For checking different parameter like torque, pressure head and velocity we used ANSYS software. Simulations in the field of hydraulic ANSYS FLUENT are used.

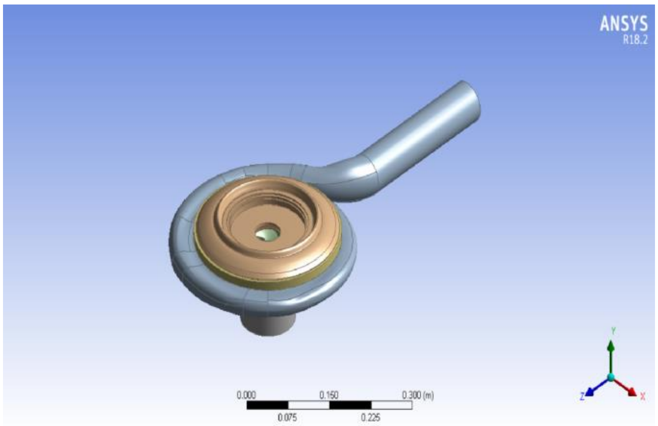

Figure 3 Computational model

C. Mesh Generation:

Mesh generation and mesh size selection includes following steps.

1) Mesh Sensitivity Test :

The process of dividing the whole component into number of elements that can be used as discrete approximation of larger domain. Mesh influences the accuracy, convergence and speed of simulation. A technique used to determine how different values of an independent variables i.e. torque, efficiency is known as mesh sensitivity test. The fluid domain is divided into number of subdivisions. The size of mesh varied to run different simulations. The values of torque, pressure, velocity are varied with mesh size. A stage reaches, although the mesh size is changed the values of torque, pressure and velocity remains same that mesh size is taken for further calculation. Below fig shows the schematic of meshing .Tetrahedron mesh element is selected for meshing. Mesh size can be varied for each and every domain. For large parts large meshing is given and for small parts fine meshing is given for better analysis.

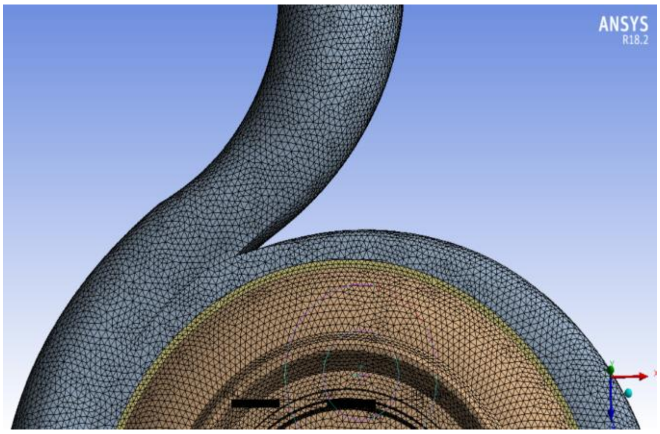

Figure 4 Tetrahedron mesh elements

2) Turbulace Sensitivity Test:

Numeriacal simulation results depends on the turbulance model used for simulation. Different turbulance models are:

1.k-epsilon:

a) Tracks changes in k : Turbulent kinetic energy.
b) Tracks changes in ε : the rate of energy dissipation i.e. kinetic energy.
c) Relatively stable and converges easily.
d) Inaccurate when simulating roating flow.

2.k- ω:

a) Tracks Specific energy disssipation.
b) Increases accuracy rotating flow, more dependent pn intial conditions.

3.Standerd k- ω: This model is great for low Reynolds no. flow with wall-bounded boundry layer.

4.SST k- ω: Has same chracteritics as k-ω. It is suitable for shear free flows as it depends on wall distacnce.

5.RNG k-ε: As it involves rapid strain it is suitable for complex shear flows.

D. Numerical simulation:

When a numerical simulation is carried out the mathematical model for given system is implemented through a computer program. Numerical simulations are important to understand the behavior of the system in different conditions. The numerical simulation was done using ANSYS CFD software various simulations were done by varying following parameters:

a) Diameter of impeller
b) Diameter of BCF
c) No. of blades

IV. RESULTS AND DISCUSSION

A. Validation of numaric results with experimental results:

In case of numerical approaches it is very important to determine the appropriateness of the model for particular application i.e. to check whether the model is right or wrong. For this purpose the results from numerical analysis are validated by comparing them with experimental results. If the proposed model is most appropriate one it gives more agreement to experimental data otherwise error goes on increasing. To have a most appropriate model the validation multiple parameters is necessary thereby validation of head, torque and efficiency is done. For validation the numerical results were obtained using model finalized after mesh and turbulence sensitivity test.

Parameters	Experimental results	Numerical results	Error (%)
Head (m)	12.33	11.12	9.81
Torque (Nm)	9.91	9.92	0.1
Efficiency (%)	65.74	72.64	9.49

Table 1 Results vs. errors

B. Effect of BCF the on PAT performance:

In experimental approach it was seen that the effect of BCF is to improve the PAT performance by reducing the disc friction losses in non-flow zone of PAT. The same effect is seen in case of numerical approach the performance of PAT was increased after the back cavity filling.

Parameters	Without BCF	With BCF	Increase
Head (m)	10.68	11.12	0.44
Torque (Nm)	9.05	9.92	0.87
Efficiency (%)	60.51	72.64	12.13%

Table 2 Change in Parameters

C. *Effect of variation of BCF diameter:*

To understand the effect of diameter of BCF on PAT performance the diameter of BCF was varied in simulation model and analysis was done. Through this analysis it is clear that variation of BCF diameter has a direct effect on performance of PAT. Performance of PAT increases with increasing BCF diameter in other words more the filling more improvement is there in the performance. The best performance point was obtained after complete filling of back cavity.

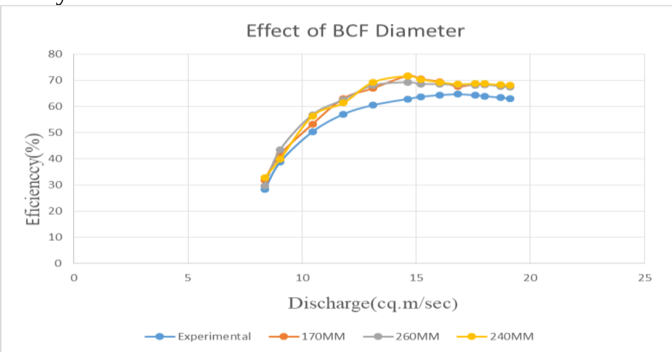

Figure 6 Efficiency vs. discharge (effect of BCF dia.)

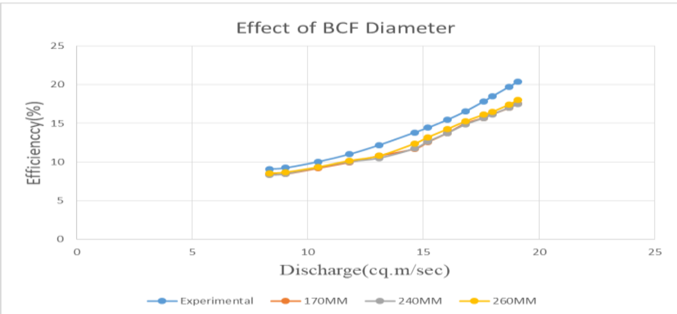

Figure 7 Head vs. Discharge (effect of BCF)

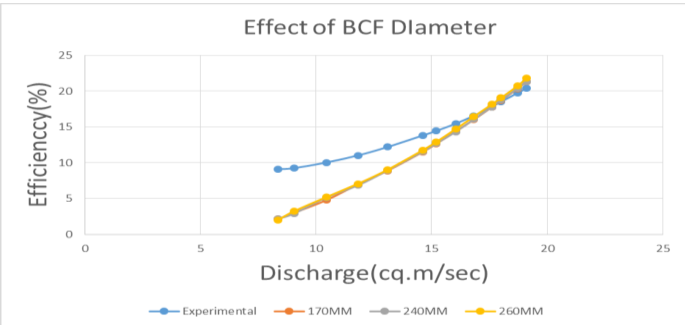

Figure 8 Torque vs. discharge (effect of BCF dia.)

V. ACKNOWLEDGMENT

We would like to express our deep and sincere gratitude to our Guide Mr. R. R. Gaji, Department of Mechanical Engineering, for guiding us to accomplish this Mega Project work. It was our privilege and pleasure to work under his able guidance.We are indeed grateful to him for providing helpful suggestion, from time to time. Due to his constant encouragement and inspiration we are able to present this mega project. We express our deep gratitude to Mr. M. M. Jadhav, Head of Mechanical Engineering Department, for his valuable guidance and constant encouragement. We are very much thankful to V. Patil, Director, Annasaheb Dange College of engineering and technology, Ashta for providing all the necessary facilities to carry out project work.

VI. REFERENCES

[1] CP Kittredge, D. Thoma, "Centrifugal pumps operated under abnormal conditions". Power (1931), pp 2022-2029.

[2] Ashish Doshi, Salim Channiwala, Punit Singh, "Inlet impeller rounding in pumps as turbines: An experimental study to investigate the relative effects of blade and shroud rounding", pp 335-347.

[3] H. H. Anderson, "Centrifugal Pumps-An Alternative Theory", pp 57-84.

[4] AT Troskolański, S Łazarkiewicz, "Energy conversions in pump and in pump systems" 1976. Barrio, R, Fernández ,Parrondo, and Blanco, (2010) "Performance prediction of a centrifugal pump working in direct and reverse mode using Computational Fluid Dynamics".

[5] Derakhshan, S., Mohammadi, B. and Nourbakhsh, A. (2009) "Efficiency Improvement of Centrifugal Reverse Pumps", Journal of Fluids Engineering, 131(2), pp. 21103.

[6] Natanasabapathi, S. R. and Kshirsagar,(2004) "Pump as Turbine - An experience with CFX-5.6."

[7] Yang, S., Kong, F., Fu, J. and Ling, X. (2012) "Numerical Research on Effects of Splitter Blades to the Influence of Pump as Turbine", International Journal of Rotating Machinery, 2012.

[8] Yang, S. S., Derakhshan, S. and Kong, F. Y. (2012) "Theoretical, numerical and experimental prediction of pump as turbine performance", Renewable Energy. Elsevier Ltd, 48, pp. 507–513.

[9] Ismail, M. A., Othman, A. K. and Zen, H. (2017) 'CFD Modelling of a Pump as Turbine (PAT) with Rounded Leading Edge Impellers for Micro Hydro Systems', MATEC Web of Conferences 87, 5004.