MATEC Web of Conferences 87, 05004 (2017) DOI: 10.1051/matecconf/20178705004
ENCON 2016

CFD Modelling of a Pump as Turbine (PAT) with Rounded Leading Edge Impellers for Micro Hydro Systems

Mohd Azlan Ismail[1], Al Khalid Othman[2], Hushairi Zen[2]

[1]*Faculty of Engineering, University Malaysia Sabah, 84500, Kota Kinabalu, Sabah, Malaysia*
[2]*Faculty of Engineering, University Malaysia Sarawak, 94300, Kota Samarahan, Sarawak, Malaysia*

Abstract. A Pump as Turbine (PAT) is one of micro hydro system components that is used to substitute a commercially available turbine due to its wide availability and low acquisition cost. However, PAT have high hydraulic losses due to differences in pump-turbine operation and hydraulic design. The fluid flowing inside the PAT is subjected to hydraulic losses due to the longer flow passage and unmatched fluid flow within the wall boundaries. This paper presents the effect of rounding the impeller leading edges of the pump on turbine performance. A CFD model of a PAT was designed to simulate virtual performance for the analysis. The aim of this study is to observe the internal hydraulic performance resulting from the changes in the performance characteristics. Highest efficiency was recorded at 17.0 l/s, an increase of 0.18%. The simulation results reveal that there is an improvement in hydraulic performance at overflow operation. The velocity vector visualization shows that there is a reduction in wake and consequently less flow separation along impeller flow passages. However, adjusting the sensitive impeller inlet geometry will also alter the velocity inlet vector and consequently change the velocity triangles for the turbo machinery system.

1 Introduction

Off-grid micro hydro power is one of most favourable types of renewable energy for generating electricity in rural areas in developing countries. Micro hydro plants with power ratings ranging from 1 to 15 kW are a normal range that can be found in most micro hydro sites. Not surprisingly, this range of power rating requires minimum technical skill to operate and maintain, eliminating professional technical expertise. Commercially available turbines such as Pelton, Crossflow, Francis and Kaplan turbines are used as the hydraulic driving mechanism. However, these types of turbines are difficult to manufacture with basic power tools and are normally imported from abroad, increasing the overall cost. The use of a pump in reverse flow, called a pump as turbine (PAT) can substitute traditional turbines, offering an affordable, feasible and practical solution for developing countries [1]. The main advantages of pumps as turbines are low capital investment and ready availability; consequently, the shorter payback period makes them suitable for self-funded micro hydro projects.

The hydraulic losses in PAT is because of long flow passage and unmatched fluid flow across the turbine as it is not designed to run in reverse flow [2]. Modifications to optimize the performance were proposed to enhance the hydraulic characteristics and consequently increase the efficiency. The modifications that were proposed include trimming the impeller blades, adjusting the blade number, adding splitter blades, installing guide vanes and rounding impeller leading edges [3-7]. Among all the modifications, rounding the impeller leading edges is the simplest method to increase PAT efficiency. This modification can be achieved by simple hand tools; grinding the impeller leading edges to a bullet shape. The purpose of rounding the impeller leading edges is to reduce the jet wake produced by sharp edges that caused flow separation.

Studies on rounding the impeller leading edges show enhanced hydraulic performance. There is an increase in efficiency between 1 and 3% [8]. A simulation model at constant speed and free vortex analysis showed a reduction in wakes at the impeller leading edges. The rounding of the impeller leading edges was found to be more beneficial for radial type flow with specific speeds less than 200 [9]. The fast flow of water along the blade passage suppresses the flow separation to a smaller region. However adjusting the sensitive inlet geometry of the impeller may change the velocity inlet vector and change the Euler momentum in the energy transfer equation [10].

The optimization process of PAT has reached the stage where internal hydraulic performance can be accomplished through a comprehensive understanding of the flow visualization. The visualization of interfaces between fluid flow, rotating impeller and pump casing can be achieved through simulation modeling. In addition, simulation modeling has the ability to investigate the fluid interaction under unsteady flow for fine adjustment, and pinpoint the root cause of the performance in the turbo machinery study. The aim of this paper is to study the effect of rounding the leading edges of impellers, on PAT performance through computation simulation modeling. This enables internal hydraulic analysis that can determine the hydraulic changes causing the variations in PAT performance.

[a] Corresponding author: lanz_mr@ums.edu.my

© The Authors, published by EDP Sciences. This is an open access article distributed under the terms of the Creative Commons Attribution License 4.0 (http://creativecommons.org/licenses/by/4.0/).

 DOI: 10.1051/matecconf/20178705004

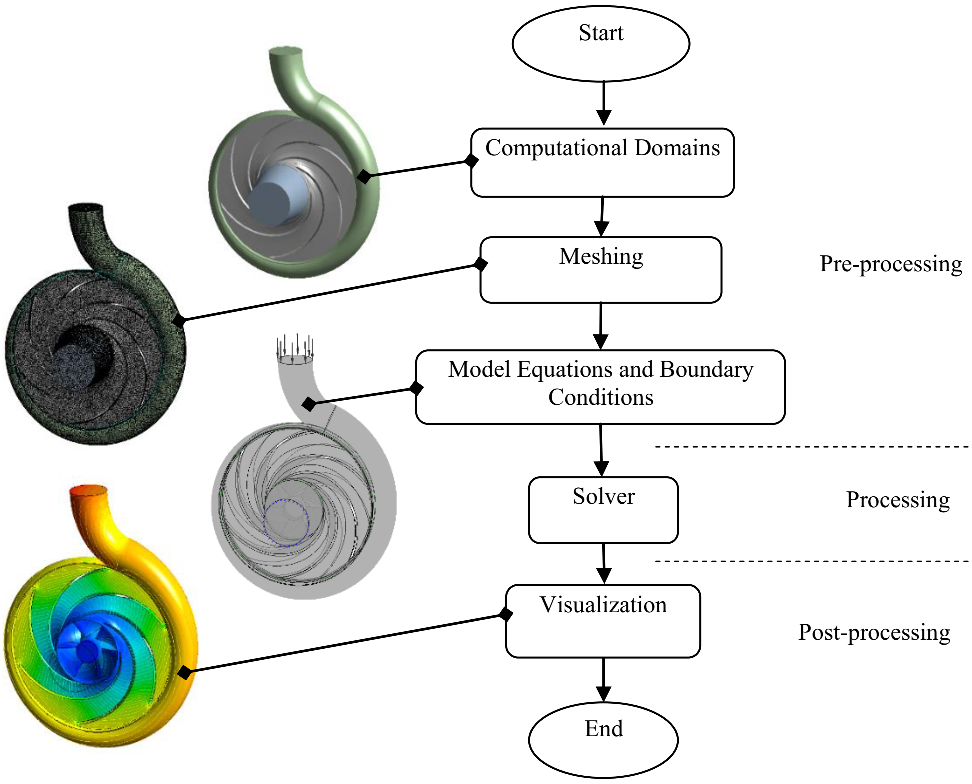
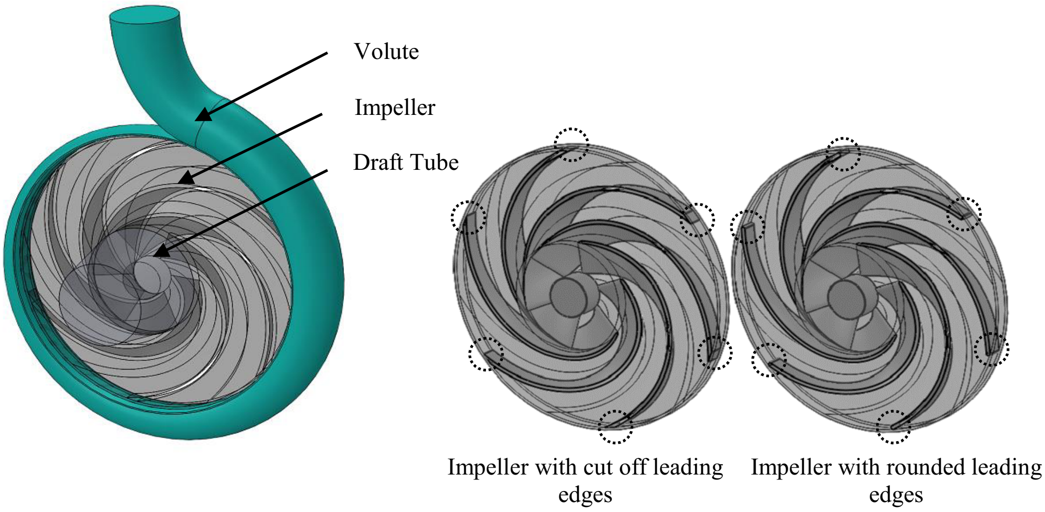


Figure 1 Flow chart of the simulation process



(a) (b)

Figure 2 (a) PAT computational domains (b) Impeller with cut off and rounded leading edges

2 Methodology

The CFD modeling workflow in this study can be categorized by three distinct stages called pre-processing, processing and post-processing. Figure 1 illustrates the chronological order of these stages.

During the pre-processing stage, the computational domains were built and the mesh was generated for the solver. The first part of this stage was to build the computational domain model that represented the wetted volume of the flow passage between PAT inlet and outlet. The processing, or solver stage, involved running all predefined parameters and codes, including the characteristics and physical properties of the system such as the fluid material properties, the physical properties of the flow, the boundary conditions, and the numerical analysis techniques. Finally, the post-processing step presented the results graphically to allow visualisation and help in thoroughly understanding and evaluating the simulation outcomes. The post-processing data was then gathered, interpreted, and discussed.

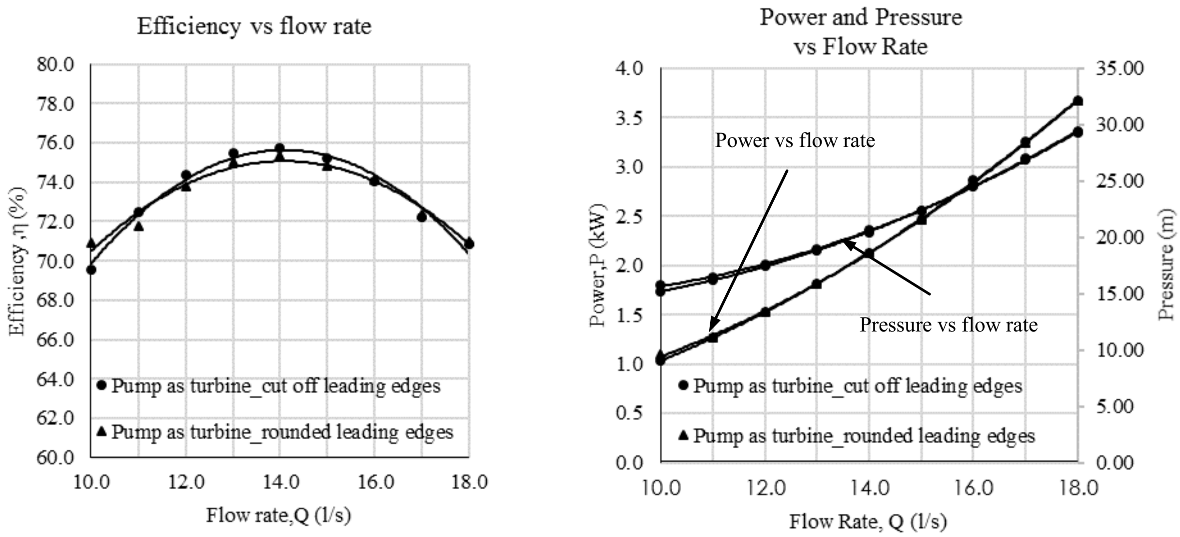


Figure 3: Simulation results for PAT with cut off and rounded leading edges

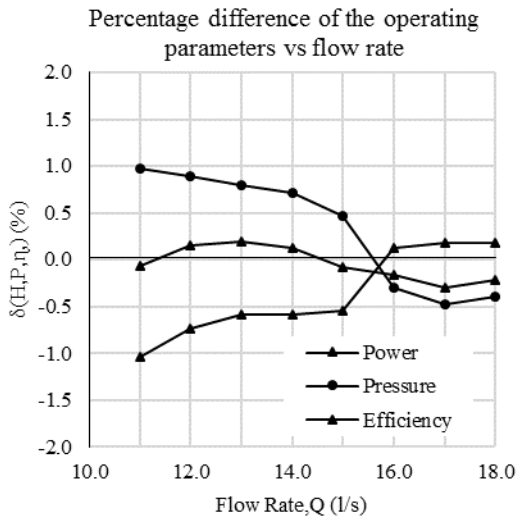


Figure 4: Percentages of difference between cut off and rounded leading edges

The computational domains of the PAT were modeled from the cavity of an end suction centrifugal pump with a specific speed of 70. Three distinct CAD parts called the volute, impeller and draft tube were designed and assembled to represent the computational domains illustrated in Figure 2 (a). The pump was disassembled and the internal dimensions were measured for the development of the CAD model. The inlet and outlet diameters were recorded at 65 mm and 50 mm, respectively. The diameter of the impeller was 214 mm and it had 6 blades. Unstructured tetrahedral mesh was chosen for the element shape, suitable for the complex geometric profile. Mesh independent analysis was carried out prior to the simulation analysis in order to determine optimum number of mesh. The total number of mesh was found at 578,892. Boundary conditions were assigned to their respective locations: inlet as mass flow rate between 10 to 18 l/s, outlet as opening to atmospheric pressure and rough wall for internal surfaces at 100 micron. The impeller was set to rotate at 1550 RPM to match the 4 pole induction generator speed.

The original impeller has the cut off leading edges profiled and modified by removing the sharp edges and rounding them to a bullet shape profile. The radius of the rounding profile was selected based on the blade thickness. In this case, the thickness of the impeller was measured at 3.0 mm and the corresponding radius of the blades was 1.5 mm. The hub and shroud of the impeller were kept in their original forms. The impeller with cut off and rounded leading edges is illustrated in Figure 2 (b).

3 Results and Discussion

The simulation results were compared with the manufacturer's data sheet to validate the accuracy of the simulation model. Table 1 presents the simulation results at the best efficient point of 8.0 l/s. The simulation model

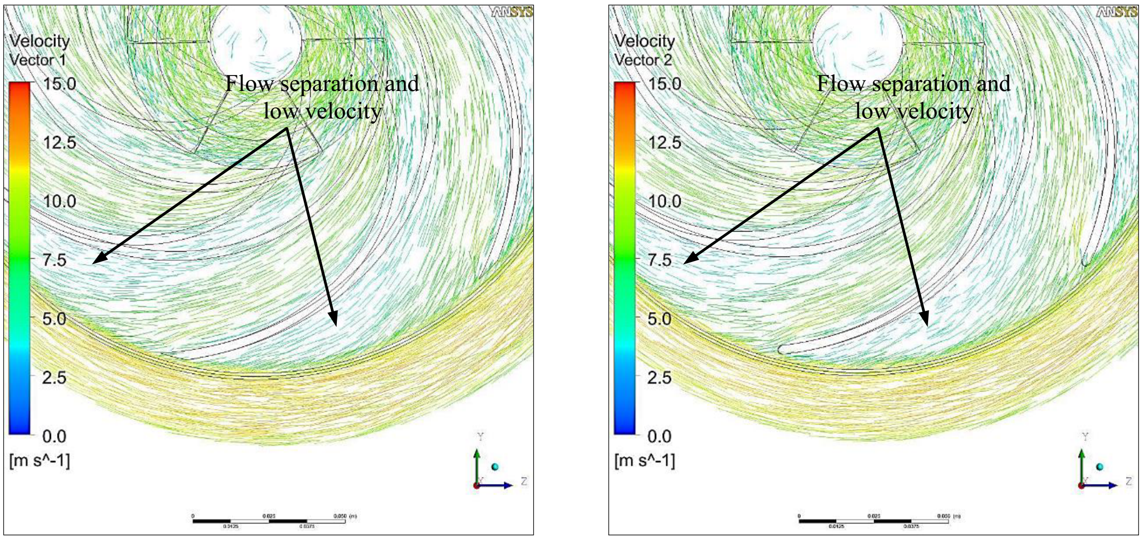


Figure 5 Velocity vector of impeller with cut off and rounded leading edges at 17.0 l/s

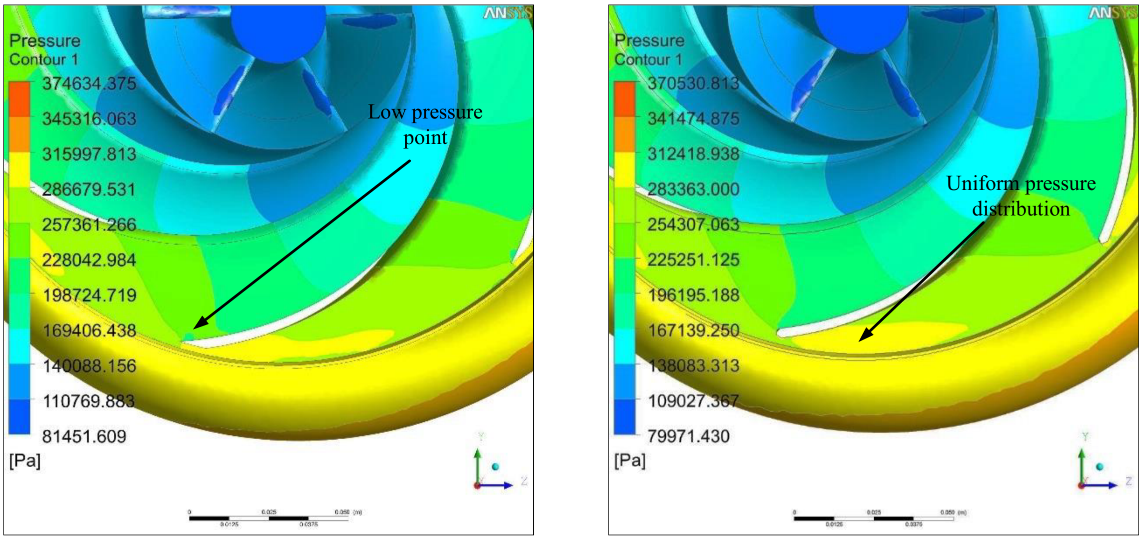


Figure 6 Pressure contours of impeller with cut off and rounded leading edges at 17.0 l/s

result yielded an acceptable level of confidence and accuracy. The lower power output generated by the rotating impeller is because of CAD model simplifications, excluding cavities between the impeller and pump housing. This simple step ignores the loss contributed by leakages and disk friction.

Table 1 Simulation Results and Manufacturer's Data Sheet at 8.0 l/s

	H(m)	P_{mech} (kW)	η(%)
Manufacturer's Data Sheet	14.0	1.70	65.0
Simulation Result	13.4	1.45	72.4

CFD modeling for the PAT performance curves is shown in Figure 3. Both impellers generated a similar performance curve with a best efficient point at 14.0 l/s. At best efficient point, the pressure of the PAT for impeller with cut off and rounded leading edges was at 20.46 m and 20.61 m, while the corresponding efficiency was 75.77% and 75.33%.

Figure 4 shows the percentages of difference of operating parameters at their respective flow rates. The efficiency differences for the rounded impeller leading edges were -1.03%, -0.73%, -0.59%, -0.58%, -0.54%, 0.13%, 0.18%, and 0.18%. The efficiency between 11.0 l/s and 15.0 l/s decreased between -0.5% and -1.1% and started to increase at 16.0 l/s, maintaining a positive percentage of difference up to 18.0 l/s. The corresponding percentages of difference in pressure were 0.98%, 0.89%, 0.79%, 0.72%, 0.47%, -0.30%, -0.48%, and -0.40%, while percentages of difference for power were -0.07%, 0.15%, 0.19%, 0.13%, -0.08%, -0.17%, -0.30%, and -0.21%.

Figure 5 shows the velocity vector inside the pump at 17.0 l/s, where the percentage of efficiency is at the highest point. It can be observed that there is a wake at the leading edges of the impeller inlet for both conditions. The flow layers detached from the blade surface and separated out from the pressure surface of the blades. The velocity vector shows the streamline is no longer tangential with the blade surfaces. The flow layer expanded from the leading edges and generated eddies. This can contribute to energy loss in the PAT and influence the flow streamlines along the impeller flow path. On the other hand, high velocity at the impeller inlet suppresses the flow separation to a smaller region. Figure 6 shows the pressure contours inside the pump at 17.0 l/s. The pressure contours for the original impeller and the modified impeller show similar pressure drop patterns across the blade passages. There is a noticeable early pressure drop region (stagnation point) at the leading edges of the cut off impeller, and non-uniform pressure variations at pressure surface of the blades.

4 Conclusion

In this paper, a simulation modeling was presented to simulate PAT performance with a rounded leading edged impeller. A CAD model of a centrifugal pump was used as computational domains. The simulation model was verified by comparing the results with the manufacturer's data sheet. The simulation result was in good agreement with acceptable replication of virtual pump performance. The efficiency of pump was recorded at 72.4%, generating 13.4 meter pressure of head at best efficient point.

Rounding impeller leading edges has negative and positive effects on the PAT performance. Removing sharp edges of the impeller reduced the wakes at the leading edges, consequently decreasing flow separation along the flow passage. Adjusting the geometry of the leading edges also changed the velocity inlet vector of the control volume. Hydraulic visualizations illustrate flow separation that leads to low velocity region along impeller flow passages. Highest decrease and increase of efficiency was achieved at part flow and over flow operation, with difference of -1.03% at 10 l/s and 0.18% at 17 l/s respectively.

It would be fruitful to pursue further research on other types of leading edge shapes such as an ellipse profile in view of increasing efficiency at all flow operations. Other types of leading edges might have different effects on velocity vectors and pressure distribution that leads to dissimilar formation of flow separations.

References

1. S. J. Williamson, B. H. Stark, and J. D. Booker, 2012. Low Head Pico Hydro Turbine Selection Using a Multi-Criteria Analysis Renewable Energy, pp. 1-8
2. S. Derakhshan and A. Nourbakhsh, 2008. Experimental Study of Characteristic Curves of Centrifugal Pumps Working as Turbines in Different Specific Speeds. Experimental Thermal and Fluid Science, vol. 32. 800-807
3. S. V. Jain and R. N. Patel, 2014. Investigations on Pump Running in Turbine Mode: A Review of the State-of-the-Art. Renewable and Sustainable Energy Reviews, vol. 30. 841-868
4. S. R. Shah, S. V. Jain, R. N. Patel, and V. J. Lakhera. 2013. CFD for Centrifugal Pumps: A Review of the State-of-the-Art. Procedia Engineering, vol. 51. 715-720
5. S. S. Yang, F. Y. Kong, X. Y. Qu, and W. M. Jiang. 2012. Influence of Blade Number on the Performance and Pressure Pulsations in a Pump Used as a Turbine. Journal of Fluids Engineering, Transactions of the ASME, vol. 134
6. Y. Sun-Sheng, K. Fan-Yu, J. Wan-Ming, and Q. Xiao-Yun. 2012. Effects of Impeller Trimming Influencing Pump as Turbine. Computers & Fluids, vol. 67, 72-78
7. V. A. Patel, S. V. Jain, K. H. Motwani, and R. N. Patel. 2013. Numerical Optimization of Guide Vanes and Reducer in Pump Running in Turbine Mode. Procedia Engineering, vol. 51, 797-802
8. P. Singh, J. Kshirsagar, S. Caglar, and F. Nestmann. 2004. Experimental and Computational Studies of the Effect of Casing Eye Rib on the Swirl Flow at the Exit of a Pump as Turbine, presented at the 2004 ASME Heat Transfer/Fluids Engineering Summer Conference, North Carolina, USA
9. S. V. Jain, A. Swarnkar, K. H. Motwanic, and R. N. Patel, 2015. Effects of Impeller Diameter and Rotational Speed on Performance of Pump Running in Turbine Mode. Energy Conversion and Management, vol. 89. 808-824
10. M. Suarda, N. Suarnadwipa, and W. B. Adnyana,, 2006. Experimental Work on Modification of Impeller Tips of a Centrifugal Pump as a Turbine. 2nd International Conference on Sustainable Energy and Environment, 21-23.