

CFD Modelling of pump as turbine with various number of blade for microhydro system

Abstract

Pump as Turbine is an electromechanically component that is largely used in microhydro system. The main advantages of pump as turbine compared to commercially available turbines are lower cost, easier to maintain and readily available. These key features make them appealing to conditions in many developing countries. However, pump as turbine has poor hydraulic performance and low efficiency thus modifications were applied to improve the performance. The aim of this paper is to study the effect of impeller blade number to the pump as turbine performance. The investigation was carried out by using commercial Computational Fluid Dynamic (CFD) software, ANSYS CFX. A centrifugal pump with a specific speed of 70 with an impeller diameter of 214.0 mm was used to generate the CAD model. The original number of the blade was at 6 and varied to 5, 7 and 8 while other geometric dimensions were kept unchanged. The simulation results reveal that the highest efficiency was attained at 7 blade number with the efficiency recorded at 76.24%. The corresponding pressure was at 20.83 meters. It was found that with increase of blade number, the efficiency and corresponding pressure of the pump as turbine also increases. However, the additional blade number reduce the cross-area flow path, consequently increase the blockage effect thus decrease the net power generated by the rotating impeller.