ABSTRACT

Experimental and numerical analysis have been carried out to evaluate and maximize the performance of a Cross Flow Turbine (CFT). CFT suits aptly for the low head and low discharge conditions, which are available in plenty and Eco-friendly but untapped. The literature scan on CFT indicates that only limited work has been done and that too are experimental. Though the design of nozzle in CFT performance is of paramount importance this was not subjected to intensive scrutiny.

An attempt has been made using Ideal Fluid Theory to evolve a suitable profile for CFT nozzle and design procedure has been established based on spiral vortex profile. The nozzle wall combinations having presently evolved spiral vortex profile and already available circular arc profiles (used by earlier investigators) are modeled numerically and analyzed using ANSYS-FLOTRAN 5.2 – a finite element analysis software. The flow through first stage runner has been analyzed numerically using the nozzle exit conditions obtained from the numerical analysis on nozzles. The different combinations of nozzle front wall and rear wall analyzed are circular profile for both walls (C-C), spiral vortex profile for both walls (S-S), and circular, spiral vortex profiles alternatively (C-S, and S-C) for the two walls.

An experimental set-up with precise instrumentation has been fabricated and the performance of CFT as an impulse turbine with single nozzle has been studied. The effect of angle of attack (16^0 and 24^0), number of blades (18, 24, and 30) and discharges (20, 30, and 45 lps) are studied under 3 m head using C-S and S-S nozzles. A concept of installing a second nozzle has been introduced to utilize the unused part of the runner periphery. The experimental investigations on CFT with two nozzles were carried out under 3-m head with flow rates of 30, and 35 lps. Experiments were carried out such that the sum of the fractional flow through the two nozzles were kept constant (either 30 or 35 lps) in order to find out the optimum flow fraction.