Influence of Nozzle Design on the Efficiency of Crossflow Turbines

Ram Chandra Adhikari, Department of Mechanical and Manufacturing, University of Calgary, Calgary, AB, Canada

David Wood, Department of Mechanical and Manufacturing, University of Calgary, Calgary, AB, Canada

Long Abstract

1. Introduction

Crossflow hydro turbines have been widely used in small scale applications for more than a century. A longstanding problem is their lower maximum efficiency when compared to more advanced turbine designs, such as Pelton and Francis. A crossflow turbine consists of two components: a rectangular curved nozzle and an impeller. The main functions of the nozzle are to accelerate the inlet flow and to provide a uniform inlet flow angle for the impeller. Analysis of the influence of nozzle configurations on the impeller performance is an important aspect of the turbine design process, but has not been investigated in any detail. This presentation will report on the main findings of the ongoing computational study on the influence of nozzle design on the efficiency. The new nozzle design is based on a simple analytic formulation which maximizes the conversion of turbine head to kinetic energy as well as providing a uniform angular momentum and radial velocity. Three-dimensional steady and unsteady RANS simulations are carried out using the commercial ANSYS CFX code in order to investigate the overall turbine performance with and without the new nozzle. The simplest multiphase model, the homogeneous multiphase model with free surface effects, was used in view of its lower computational time and satisfactory results for such flows. The SST $k - \omega$ turbulence model was used as this model has been shown to have better performance for turbomachinery computations compared to other two-equation models and its superior performance in highly separated flows [2], [3]. Potential efficiency gain is analyzed based on the flow field characteristics at the nozzle exit or the impeller inlet. The main results are summarized below.

2. Results and Discussion

To assess the capability of RANS/URANS solutions or the validity and the accuracy for simulating the turbine performance, the measured power outputs at different flow rates, head and impeller speeds for a 7 kW turbine reported by Dakers and Martin [1], were compared with the CFD results. The RANS/URANS solutions were then used to examine the velocity, the angular momentum and the angle of attack at the impeller inlet for the existing nozzle and the new nozzle design.

Figure 1 shows the comparison of experimental and the CFD predicted results for the turbine power output over a range of impeller speeds. The RANS/URANS results for the power outputs were found within 7% of the experimental results as shown in Figure 1. Figure 2 shows the typical flow patterns in the turbine at maximum efficiency points. The tangential and radial velocities and angle of attack for the existing and modified nozzle designs at the impeller inlet are summarized in Figure 3 and 4. The Figures show that the angular momentum and angle of attack are not uniform for the existing nozzle (at maximum efficiency point), whereas the angular momentum for the new nozzle design is uniform as well as increased. Also the angle of attack is more uniform. The new nozzle differs from the existing nozzle in the size of nozzle throat and rear wall shape. This is the key reason why the new nozzle design has improved the turbine efficiency. Improvement in power outputs for the existing and new nozzles at different impeller speeds are summarized in Table 1.

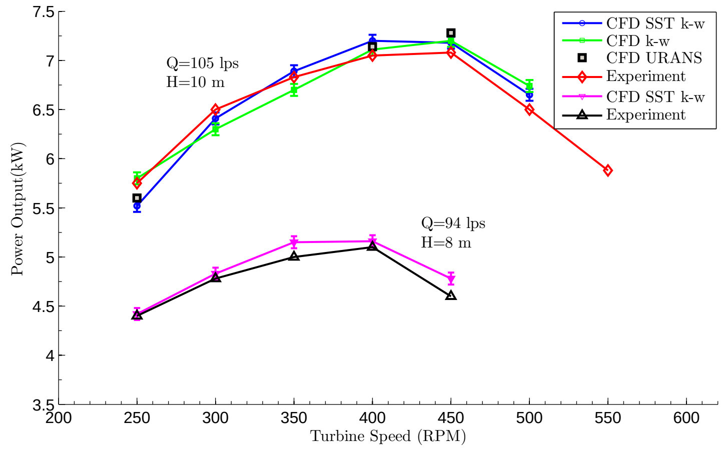
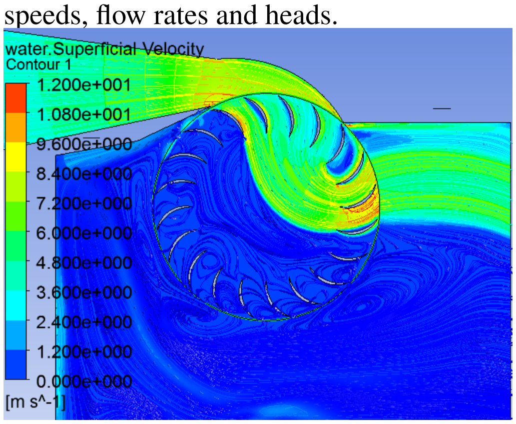
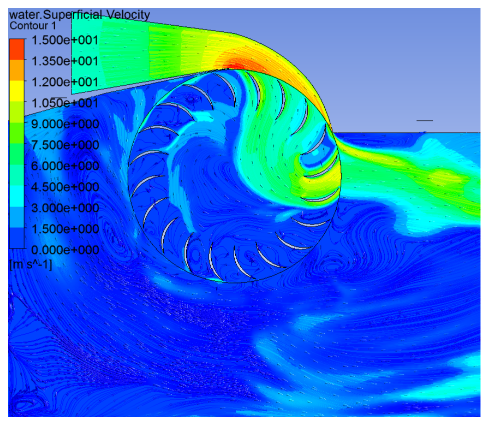


Figure 1. Comparison of CFD predicted and measured turbine power outputs at different impeller speeds, flow rates and heads.



(a) Existing nozzle design (450 RPM)



(b) New nozzle design (500 RPM)

Figure 2. Water velocity contours with streamlines superimposed for two nozzle designs at maximum efficiency.

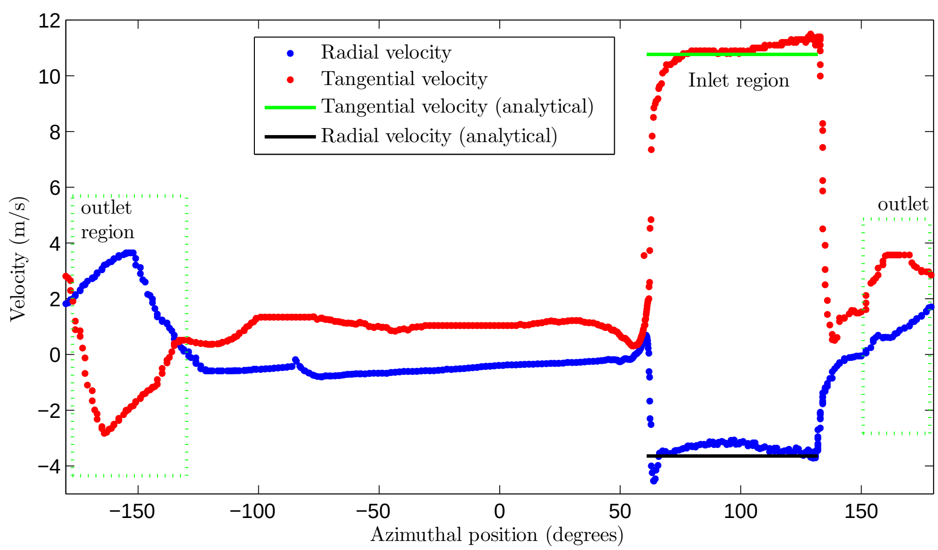


Figure 3. Comparison of radial and tangential velocities (CFD and analytical) at the impeller boundary for the existing nozzle

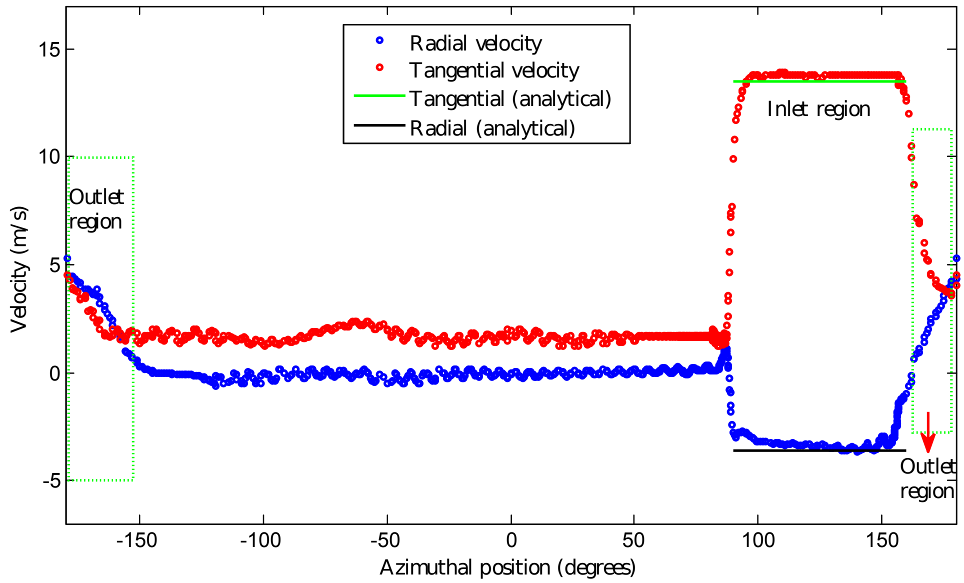


Figure 4. Comparison of radial and tangential velocities (CFD and analytical) at the impeller boundary for the new nozzle

Table 1. Comparison of turbine power outputs for two nozzle designs

Impeller Speed (RPM)	Existing Nozzle (kW)	New Nozzle (kW)
450	7.18	7.20
500	6.53	7.87
550	6.04	7.66

3. Conclusion

The SST $k - \omega$ with homogeneous multiphase and free surface effects model showed a satisfactory agreement with the measured power for a small scale crossflow hydro turbine. Using this computational model, it was shown that nozzle configuration has a significant effect on the impeller performance and needs detailed examination in order to match the impeller design. With the new nozzle design method, it is possible to maximize the conversion of potential head to kinetic energy, obtain a uniform angular momentum and angle of attack at the impeller inlet. For this, nozzle throat, rear wall shape and entry angle can be optimized for the given flow rate and head. A detail computational analysis would help determine an optimum nozzle design; the results of the detail analysis will be presented in the conference.

References

[1] Dakers A.J. and Martin G. Development of a simple cross-flow water turbine for rural use. *Conference on Agricultural Engineering, Armidale*, pages 32–42, 1982.

[2] Menter F. R. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA-Journal*, 32(8):1598 – 1605, 1994.

[3] Denton J.D. Some limitations of turbomachinery cfd. *Proceedings of ASME Turbo Expo 2010: Power for Land, Sea and Air, Glasgow, UK*, 32(8):1598 – 1605, June 14-18, 2010.