Numerical Investigation of Cavitation Improvement for a Francis Turbine

Zhifeng Yao[1,2], Ruofu Xiao[1,2], Fujun Wang[1,2*], Wei Yang[1,2]

[1]College of Water Resources and Civil Engineering, China Agricultural University, Beijing 100083, China
[2]Beijing Engineering Research Center of Safety and Energy Saving Technology for Water Supply Network System, Beijing 100083, China
E-mail: wangfj@cau.edu.cn

Abstract. Cavitation in hydraulic machine is undesired due to its negative effects on performances. To improve cavitation performance of a Francis turbine without the change of the best efficiency point, a model runner geometry optimization was carried out. Firstly, the runner outlet diameter was appropriately increased to reduce the flow velocity at runner outlet region. Then, to avoid the change of the flow rate at the best efficiency point, the blade shapes were carefully adjusted by decreasing the blade outlet angles and increasing the blade wrap angles. A large number of the modified runners were tested by computational fluid dynamic (CFD) method. Finally the most appropriate one was selected, which has the runner outlet diameter 10% larger, the blade outlet angles 3 degrees smaller and the blade wrap angles 5 degrees larger. The results showed that the critical cavitation coefficient of the model runner decreased at every unit rotational speed after the optimization, and the effect was much remarkable at relative high flow rate. Besides, by analysing the internal flow field, it was found that the zone of the low pressure on pressure surface of the optimized turbine blades was reduced, the backflow and vortex rope in draft tube were reduced, and the cavitation zone was reduced obviously.

1. Introduction

Cavitation can occur within the entire operating conditions of hydro turbines, once the absolute static pressure at a certain location falls below the saturated vapor pressure of the fluid at the prevailing temperature conditions [1]. The bubbles condense when they reach a region of higher pressure and finally collapse or implode. Cavitation in hydraulic machinery may lead to flow instabilities, excessive vibration, damage to material surfaces and degradation of machine performance [2]. Nowadays, cavitation issue is becoming more and more important, because the operating speeds are increasing as the results of the output power growth and reduction of geometry dimensions to decrease the cost of its components. Also it is subjected to operate modern turbines in the conditions far from the best efficiency point, and the cavitation phenomena are prone to occur at off design operating conditions [3].

For a Francis turbine, one of the well-known cavitation phenomena is the draft tube whirl, which is a cavitation vortex-core flow formed just below the runner cone in the center of the draft tube [2]. Müller et al.[4] gave an introduction to the cavitation in hydraulic machinery. He pointed out that the whirl cavitation developed from the hub of the runner to the center axis of the draft tube in the bulk flow. The vortex rotates in the same direction as the runner at part load and in the opposite direction at overload. In low flow rate regimes, the vortex core takes a helical shape and rotates at a speed between 0.25 – 0.40 times the runner rotational frequency. The whirl development usually is the main source of pressure fluctuations in hydraulic installation [5]. Alligne et al [6] predicted the cavitation surge phenomenon in the draft tube. It is shown that convective terms have a stabilizing influence modifying stability limit prediction driven by the divergent geometry modelling of draft tube. Wu et al. [7] applied numerical simulation and model test to investigate the cavitation flow in Francis turbines. Simulation results showed reasonable agreement with the experiment data by using the *SST k- ω* turbulence model.

In present paper, flow passage geometry of a Francis turbine was optimized to improve the cavitation performance. CFD method was employed to uncover the detail of the flow field inside the runner and draft tube. And the cavitation characteristics before and after optimization were compared.

2. Model runner and optimization procedure

The original runner was a reduced scale model having 16 stay vanes and 24 guide vanes. The height of the guide vane was 54mm. The runner inlet diameter D_1 was 410 mm and the outlet diameter D_2 was 280 mm. The runner had 15 long blades and 15 short blades. Figure 1 shows the three dimensional model of original runner.

The optimization procedure mainly included two steps. Firstly, the runner outlet diameter was increased by 3% to 12% of the original outlet diameter. The cavitation performance may improve due to the flow velocity decrease at the runner outlet region. But the flow rate at the best efficiency point will definitely changed, and also the efficiency performance may decline. In order to avoid such negative effects, the blade shapes were carefully adjusted by decreasing the blade outlet angles around 2 to 4 degrees and increasing the blade wrap angles 2 to 7 degrees.

A large number of the modified runners were tested by computational fluid dynamic (CFD) method. Finally the most appropriate one was selected. Figure 2 shows the comparison of meridional shapes of original and optimized runners. The blue line shown in Figure 2 is the optimized runner, and the black line is the original one. The outlet diameter of the optimized runner was 1.1 times of the original one, while the inlet diameter D_1 was kept the same. The blade of the runner should be adjusted properly due to the increase of outlet, and the best efficiency point was also kept the same with the original one. The final blade outlet angles of the runner after optimized were 3 degrees smaller than the original one. The blade wrap angles were 5 degrees larger than the original one. The inlet of draft tube was increased to match the outlet of optimized runner with the shape lines unchanged.

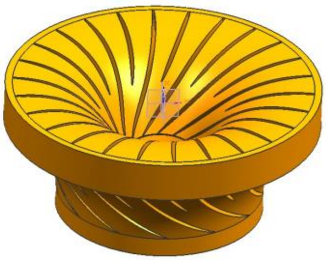


Figure 1. Original runner

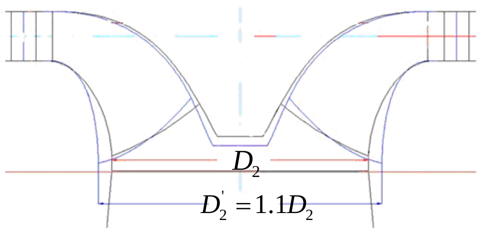


Figure 2. Comparison of meridional shapes of original and optimized runners

3. Numerical simulation method and meshing

The Francis turbine model used in CFD mainly consisted of five components, such as spiral volute, stay vane, guide vane, runner and draft tube. The flow domains were meshed in ICEM. The total mesh element quantity was 1,270,700 after grid-independent analysis. Structured meshes were generated to define the daft tube, stay vane, guide vane, runner and spiral volute around 138,200, 99,300, 168,200 and 572,500, respectively. For the spiral volute, unstructured mesh was applied with 292,500 elements, and a mesh refinement was used near the tongue zone.

The flow in Francis turbine was considered to be incompressible, viscous and fully turbulent. The Reynolds averaged Navier–Stokes (RANS) approach with *SST k-ε* turbulent model was used to simulate the steady flow by using the commercial code CFX. The Rayleigh-Plesset cavitation model was used to capture the cavitation phenomena. The wall functions based on the logarithmic law were adopted to model the boundary layers. The PISO algorithm was used to calculate pressure velocity coupling. High resolution scheme was used for advection terms and turbulence numeric. Frozen rotor method was adopted to simulate the relative motion of the runner grid with respect to the suction and volute. For a specific operating condition, the total pressure boundary condition was established at the inlet, and the average pressure boundary was imposed at the outlet of the computational domain. The volute and draft tube walls were set in stationary frame and modeled using no-slip boundary conditions.

4. Results and Discussions

4.1. Comparison of efficiency

Figure 3 shows the comparison of efficiency between the optimized and original runners. At relative high unit rotational speed, the efficiency of optimized one is a little bit higher than the original one at the same guide vane opening. While at relative small unit rotational speed, the efficiency of optimized one is lower than the original one. However, the largest difference is lower than 1%. Figure 4 shows the comparison of the pressure distribution along the blade surface flow line at 90% span near bottom ring (Guide vane opening is 18°, n_{11}=62.5 r/min, P_{tout}=20 kPa.). The pressure imposed on the blade surface close to runner outlet for the optimized one is much larger than the original one.

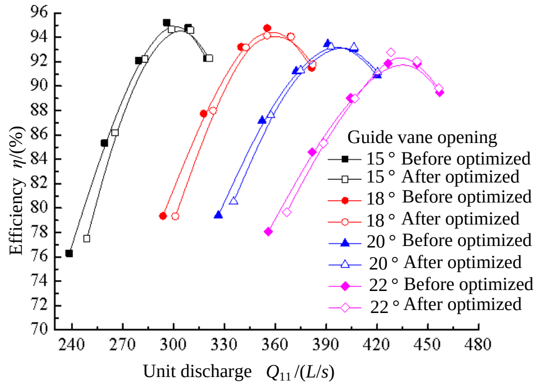


Figure 3. Comparison of efficiency

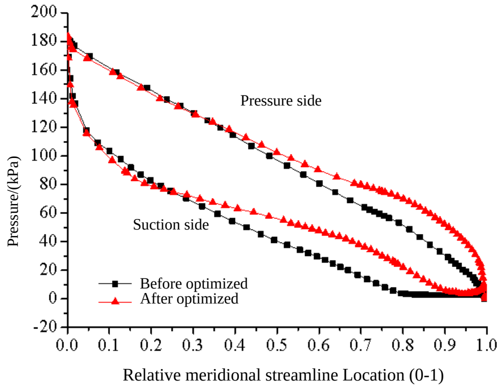
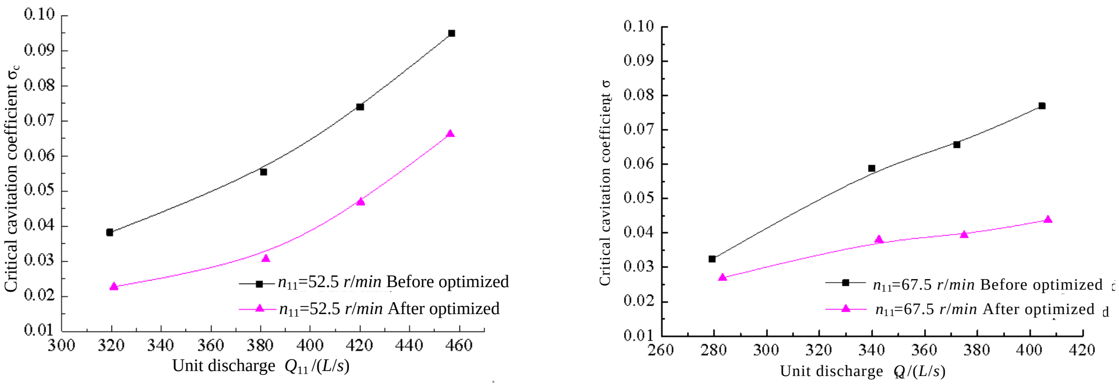


Figure 4. Comparison of the pressure distribution on blade surfaces

4.2. Comparison of cavitation

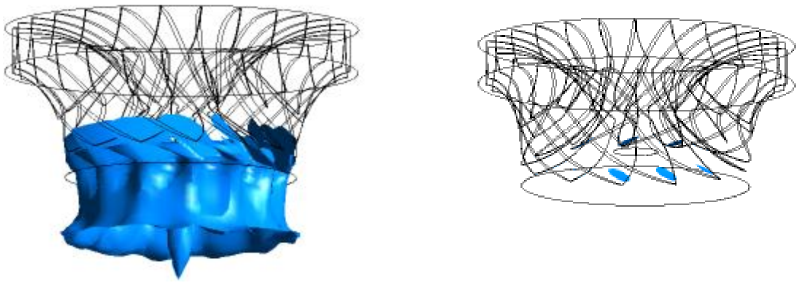
The cavitation coefficient was defined as $\sigma = {p_{tout}\text{-}p_v}\big/{\rho g H}$. where P_{tout} was the total pressure of the outlet of draft tube, Pa. P_v was the saturated vapor pressure at specific temperature, Pa. ρ was the density of water, kg/m^3. g was the acceleration of gravity, m/s^2. H was the head of the turbine, m. The specific cavitation coefficient corresponding to the 1% drop of efficiency was defined as the critical cavitation coefficient σ_c. Figure 5 presents the comparison of critical cavitation coefficient when the unit rotational speeds were 52.5 *r/min* and 67.5 *r/min*. It is quite obvious that runner optimization has greatly decreased the critical cavitation coefficient of original runner. And the effect is much remarkable in relative high flow rate.



(a) n_{11}=52.5r/min (b) n_{11}=67.5r/min

Figure 5. Comparison of critical cavitation coefficient

Figure 6 shows the comparison of gas volume fraction in runner and draft tube with the isosurface value equaling to 10%. The large cavitation regions appear in runner outlet and draft tube for the original one. While only small cavitation regions on the blade close to outlet are discovered in the optimized runner. The backflow and vortex rope in draft tube are reduced. Increasing runner outlet diameter can reduce the flow discharge velocity and thus increases the static pressure at the runner outlet contributing to positive effect on cavitation. Therefore, the cavitation performance is greatly improved after optimization.



（a） Before optimization （b） After optimization

Figure 6. Comparison of vapour volume fraction in runner and draft tube (Isosurface value=10%, guide vane opening: 18°, unit rotational speed n11=62.5 r/min)

5. Conclusions

To reduce the flow velocity at the runner outlet region of the original model runner, the diameter of runner outlet was enlarged. Combining with the adjustments of the blade shapes by decreasing the blade outlet angles and increasing the blade wrap angles, the best efficiency point was kept the same. After a large number of modification runner test by CFD method, the final optimized one was selected. The optimized runner had the outlet diameter 10% larger than the original one. The blade outlet angles were 3 degrees smaller, and the blade wrap angles were 5 degrees larger than the original ones. The critical cavitation coefficients of the optimized runner decreased at every tested unit rotational speed comparing with the original one, especially at the relative large guide vane opening conditions. The zone of the low pressure on pressure surface of the optimized runner blades was reduced, which had the positive effect on cavitation. Proper increase the outlet of the Francis runner combined with adjustments of the blade shapes may have benefits on improving cavitation performance without changing the efficiency.

Acknowledgement

The authors would like to acknowledge the financial support given by the National Natural Science Foundation of China (No. 51139007, 51409247) and the National Science & Technology Support Project of China (No. 2012BAD08B03, 2015BAD20B01).

References

[1] Kumar P, Saini R P. Study of cavitation in hydro turbines--A review. Renewable & Sustainable Energy Reviews, 2010, 14(1): 374-383.
[2] Escaler X, Egusquiza E, Farhat M, Avellan F, Coussirat M. Detection of cavitation in hydraulic turbines. Mechanical Systems and Signal Processing, 2006, 20(2006): 983-1007.
[3] Hasmatuchi V, Farhat M, Roth S, Botero F, Avellan F. Experimevidence of rotating stall ina pump-turbine at off-design conditions in generating mode. Journal of Fluids Engineering, 2011, 133(5): 623-635.
[4] Müller A, Dreyer M, Andreini N, et al. Draft tube discharge fluctuation during self-sustained pressure surge: fluorescent particle image velocimetry in two-phase flow. Experiments in fluids, 2013, 54(4): 1-11.
[5] Müller A, Favrel A, Landry C, et al. On the physical mechanisms governing self-excited pressure surge in Francis turbines //IOP Conference Series: Earth and Environmental Science. IOP Publishing, 2014, 22(3): 032034.
[6] Alligne S, Nicolet C, Tsujimoto Y, Avellan F. Cavitation surge modelling in Francis turbine draft tube. Journal of Hydraulic Research, 2014, 52(3): 399-411.
[7] Wu Y, Liu J, Sun Y, Liu S, and Zuo Z. Numerical analysis of flow in a Francis turbine on an equal critical cavitation coefficient line. Journal of Mechanical Science and Technology, 2013, 27(6): 1635-1641.