ANALYSIS OF THREE-DIMENSIONAL FLOW THROUGH A CONCRETE SPIRAL CASE

ZHENG XIAOBO, LUO XINGQ

Xi'an University of Technology Xi'an 710048
(e-mail address: zhengxbb@163.com)

The scroll case is an important component of the water turbine. To study three-dimensional flow through the scroll case is of great benefit to improve the performance of the turbine. It will cost much manpower, material resources and much money to obtain a satisfied analysis result of the scroll case through experiment. Furthermore, the experiment will spend long time. Accordingly, the analysis of flow field within the spiral case with numerical simulation method has been a main tool to optimize the spiral case.

With the rapid development of computational method and computer, numerical simulation method and CFD method have been widely used in hydraulic machine field, especially in the analysis of flow field within the inlet and outlet of the water turbine [1-6]. Based on the Reynolds-averaged N-S equations and the standard k-ε model, the numerical simulation of three-dimensional flow through a concrete spiral case was conducted by using the SIMPLE scheme with the non-structural grid systems in the paper. The analysis for the flow through concrete spiral case with one pier and two piers was present. The effect of the piers on the distribution of velocity and press in the spiral case, outlet-angle, efficiency of the spiral was studied.

According to the results, compare with the one-pier scheme, the area of flow section was decreased because of the increase of the pier number in two-pier schemes, thus to increase the velocity, and the enhanced hydraulic friction caused the decrease of the efficiency.

Because the piers limited the flow direction of part water, the outlet discharges distribution of spiral case tend to uneven, and the outlet-angle distribution of spiral case became unsteady. However, in some project, when more piers are needed because of larger inlet width of the spiral case, the outlet discharges and outlet-angle distribution can be bettered by optimizing the location and the head shape of the piers, thus can increase the spiral case efficiency.

REFERENCES

Weili Liao, Li Jianzhong. Numerical Simulation and Experimental Investigation of Flow in End Clearance Region of Guide-Vane Row of Hydraulic Turbine, Proceedings of the Hydraulic Machinery and Systems *21th IAHR Symposium Sept.12-16, 2002, Lausanne.* pp: 43-51.

LIAO Wei-li, LI Jian-zhong. Numerical Simulation and Study of Internal Flows Through the Spiral Case, Journal of Xi'an University of Technology, Vol.18 No.1, 2002.1

Yu Bo, Flow Field Simulation of High Narrow Turbine Draft Tube, Journal of Yangtze River Scientific Research Institute, Vol.16 No.6,1999.12

Agouzoul M, Reggio M, Camarero R. Calculation of Turbulent Flows in a hydraulic Turbine Draft Tube. Journal of Fluids Engineering, 1990(9)