Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and 8th International Conference on Nanochannels, Microchannels, and Minichannels
FEDSM-ICNMM2010
August 1-5, 2010, Montreal, Canada

FEDSM-ICNMM2010-30283

Numerical Simulation and Experimental Study of the Pressure Fluctuation in Water Turbines

Ru-Zhi Gong[1], Hong-Jie Wang[1], Wan-Jiang Liu[2], Da-Qing Qin[2], Feng-Chen Li[1],

[1] School of Energy Science and Engineering, Harbin Institute of Technology, Harbin 150001, P.R China

[2] Harbin Institute of Large Electrical Machinery, Harbin 150001, P.R China

ABSTRACT

Pressure fluctuation is a common problem in large-scale hydraulic turbine, which will affect the performance of the water turbines, such as negatively affecting the efficiency, increasing the damage of related components and decreasing the life span, bringing up great potential troubles to the operation safety of large-scale water turbines. The mechanism how the pressure fluctuation appears is due to the low-frequency pressure pulsation in the draft tube and the mid-frequency pressure pulsation generated by dynamic and static interferences before the runner transmit upstream, and the low-frequency and mid-frequency pressure pulsation in the front of the runner, guide vanes, fixed vanes and the inlet of the volute.

To study the relationship between the capacity and the pressure fluctuation of the water turbine, the CFD calculation was performed on the water turbine and the results of the calculation were compared with the measurements in this study. The model of water turbine studied in this paper is Baihetan HEC_1014-type. The computations were carried out in the whole runner blade passage and the boundary conditions were set as the same as the experimental conditions. The unsteady state flow of the turbine was computed. The effects of turbulence were modeled with standard κ-ε turbulence model. The inner flow field and the pressure fluctuation were obtained from the calculation using the solver of Fluent. And the results of simulation are compared with the experimental results.

Keywords

water turbine; numerical simulation; pressure fluctuation; whole runner blade passage

Introduction

The flow in water turbines has been investigated numerically and experimentally for decades. There are 1D, 2D and 3D numerical methods that simulate the flow in turbines. The flow conditions, analyses of the inner flow of turbines, and the way to improve the capacity of turbines are researched in the past.

Pressure fluctuation is a common problem in large-scale hydraulic turbine, which will affect the performance of the water turbines, such as negatively affecting the efficiency, increasing the damage of related components and decreasing the life span, bringing up great potential troubles to the operation safety of large-scale water turbines. The mechanism how the pressure fluctuation appears is due to the low-frequency pressure pulsation in the draft tube and the mid-frequency pressure pulsation generated by dynamic and static interferences before the runner transmit upstream, and the low-frequency and mid-frequency pressure pulsation in the front of the runner, guide vanes, fixed vanes and the inlet of the volute.

Bian[1],Li[2]and Choo[3] respectively gave methods to simulate the flow field inner the centrifugal hydrolyc machines,。 Water turbines were studied experimentally by Kergourlay, Kouidri et al.[5] and Chen, Tian[6]. Nilsson, Davidson[7] and Sultanian, Nagao et al.[8] validated the simulation results with the measurements, and proved that the numerical methods for simulating the inner flow of water turbines are credible. White, Holloway and Gerber[9] studied certain water turbine and gave the numerical method to pridict the performance of the turbine with the help of CFD.

The computational cases

The turbine studied in the paper is the BaiHeTan water turbines. The model has 15 runner blades, 23 stay vanes and 24 guide vanes with a runner radius of R_{ref}=0.42m. The all detailed geometrical information of the turbine, including vanes, guide vanes, runner and draft tube were obtained from Harbin Institute of Large Electrical Machinery. The model is shown as Fig.1.

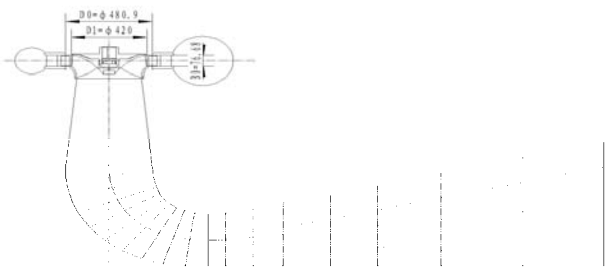

Fig.1 the geometry of BaiHeTan turbine model

The model has been experimentally tested in Mar. 2009. Several off-design conditions have been studied in present work, Tab.1 shows the operating condition studied in this work. Here n_{11}[rpm] is the specific runner angular rotational velocity, H[m] is the water head.

Tab.1 the operating conditions studied

Operating condition	H [m]	n_{11} [rpm]
1	115	84.42

2	130	79.47
3	170	69.47

Numerical considerations

The flow in water turbines can be described with the equations below. The steady Reynolds time-averaged continuity and Navier-Stokes equations for incompressible flow read

$$\frac{\partial u_i}{\partial x_i}=0$$

$$\frac{\partial u_i}{\partial t}+u_j\frac{\partial u_i}{\partial x_j}=\frac{1}{\rho}\frac{\partial}{\partial x_j}\left[-p\delta_{ij}+\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k}\right)+\left(-\rho\overline{u_i'u_j'}\right)\right]$$

Where

$$-\rho\overline{u_i'u_j'}=\mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\left(\rho k+\mu_t\frac{\partial u_i}{\partial x_i}\right)\delta_{ij}$$

μ_t is turbulent viscosity, k is turbulence kinetic energy, $k=\overline{u_i'u_j'}/2$.

The medium in the water turbine is water-liquid and the flow is not too fast, the flow in the turbine is incompressible. The influence of viscosity can be considered by the standard k-ε turbulence model. The standard k-ε turbulence model is described as the following equations

$$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho u_i)}{\partial x_i}=0$$

$$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial(\rho u_i u_j)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\mu_c\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right]-\frac{\partial P^*}{\partial x_i}+f_i$$

$$\mu_t=\rho C_u\frac{k^2}{\varepsilon}$$

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho u_i k)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\left(\frac{\partial k}{\partial x_j}\right)\right]+\rho(p_r-\varepsilon)$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho u_i\varepsilon)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\left(\frac{\partial\varepsilon}{\partial x_j}\right)\right]+\frac{\rho}{k}\left(C_1\varepsilon p_r-C_2\varepsilon^2\right)$$

Where μ_c is the effective viscosity coefficient, it can be described as the summary of the molecular the viscosity coefficient and the turbulence viscosity coefficient; p_r is the source of turbulence kinetic energy, read

$$p_r=\frac{\mu_t}{\rho}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j}$$

The standard k-ε turbulence model can ascertain the distributions of various complex length because the standard k-ε turbulence model considered the transition of the turbulence velocity and the transition of the turbulence length. At the same time, the basic form of the standard k-ε turbulence model is simple, and it can describe flow and reversed flow in shear layer. k-ε turbulence model can describe the physical process of kinds of flow authenticly. It has the charictoristics of commonality, fine precision and little calculational quantity.

The inlet boundary condition is given as the experiments. Pressure-inlet and pressure-outlet boundary conditions were set for the inlet and outlet of the turbine respectively. The Gambit grid generator is used for grid generation. To improve the precision of the calculation, the grid in the runner is dense and the grid outer the runner is relatively sparse. There are 2.86 million cells for the simulation in the whole model, Fig.2 is the sketch map of the grid in the turbine model. And atter the calculation was done to the turbine model, Matlab is used for the help of post-processing.

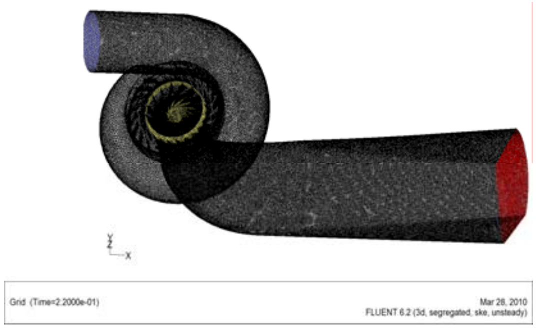

Fig.2 sketch map of the grid in the model

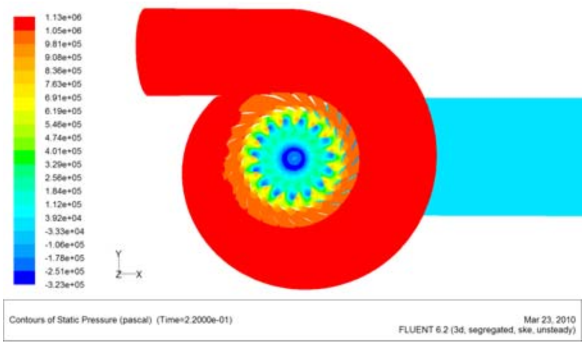

Fig.3 the whole pressure distribution of the model

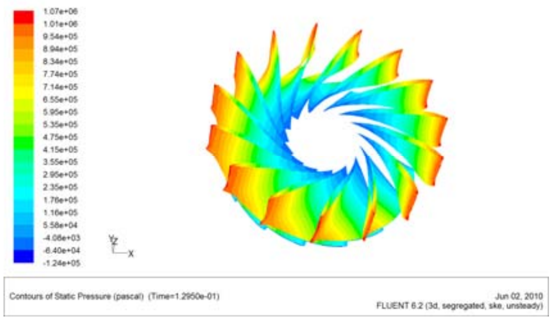

Fig.4 the pressure distribution on the blades

4. DISCUSSIONS

After the simulation the flow field inner the turbine was obtained. The whole pressure distribution of the model is shown in Fig.3. And the pressure distribution on the blades is shown in Fig.4.

After the simulation, the same position as that in the experimental model was chosen to compute the pressure fluctuation of the water turbine, so as to compare the results of simulations with the results of experiments. Fig.5-Fig.7 show the comparisons between computational results and experimental results. They show the relationship between the pressure fluctuation and the power of the turbine both the simulation results and computational results.

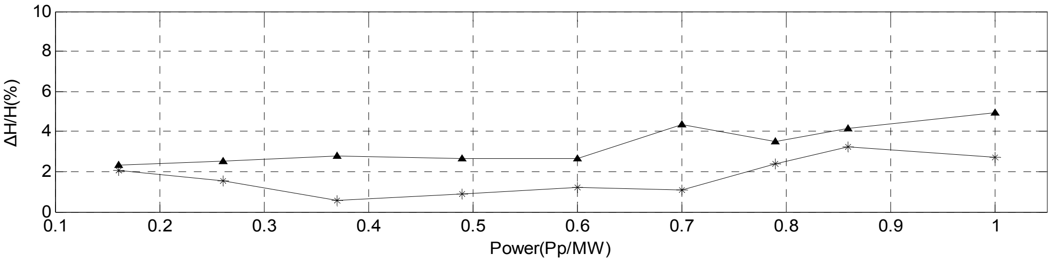

Fig.5 The contrast of numerical results and experimental results under the condition of Hp=115m

——*——numerical results ——▲——experimental results

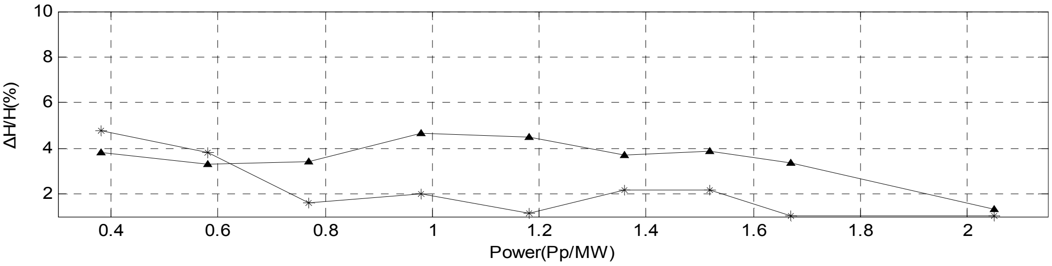

Fig.6 The contrast of numerical results and experimental results under the condition of Hp=130m

——*——numerical results ——▲——experimental results

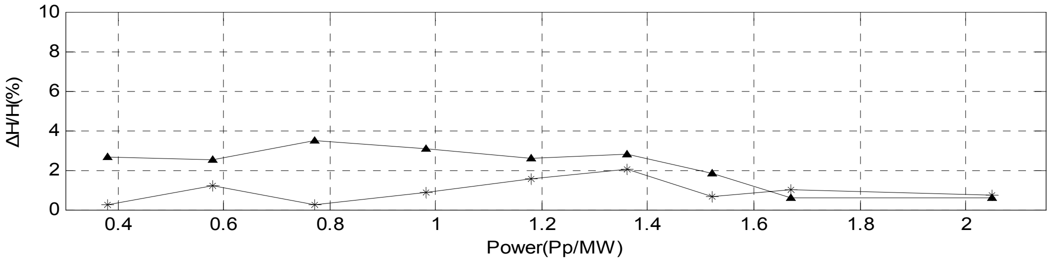

Fig.7 The contrast of numerical results and experimental results under the condition of Hp=170m

——*——numerical results ——▲——experimental results

The figures above show that the trend of the computational results is consistent with the trend of experimental resuts. But the there are little difference between the computational results and experimental results. The curves of the computational results is under the curve of experimental ones. This maybe caused by the different measuring methods. The data of measurements is acquired by the pressure sensor on the measuring point, but the results of the simulation are acquired through average the pressure in certain area.

Conclusion

The present work simulated the flow inner the BaiHeTan1014-type water turbine model. The pressure fluctuation of the turbine is computed and analyzed. The simulation results are validated against with the measurements. The unstructured grid was used in the calculation, and the standard κ-ε turbulence model was used to take the reflection of turbulent flow into account. The results of simulation is in acord with the experimental results in the trend. Because of the difference between the simulations and experiments in data acquisition, there are little diversity between the computational results and experimental results.

References

[1] Bian, XiaoDong Tang, XuDong Huang, DongTao Tian, ManCang et al Application of Numerical Simulation of the Flow Field of Centrifugal Impellers for Fan Design, TSINGHUA SCIENCE AND TECHNOLOGY, Volume 5 , Number 1 , March 2000,82-88

[2] Li Yi-min, Zhou Zhong-ning Investigation and numerical simulation of inner-flow of an axial mineflow fan under low flow rate conditions, J China Univ Mining & Technol 18 (2008) 0107–0111

[3] Yin Chin Choo, Kashem M. Muttaqi, Michael Negnevitsky, Modeling of Hydraulic Turbine for Dynamic Studies and Performance Analysis,

[4] C. J. Meyer, and D. G. Kroger, Numerical simulation of the flow field in the vicinity of an axial flow fan, International Journal For Numerical Methods In Fluids *Int. J. Numer. Meth. Fluids* 2001; 36: 947–969

[5] Gerald Kergourlay, Smaine Kouidri, Gary W. Rankin, Robert Rey, Experimental investigation of the 3D unsteady flow field downstream of axial fan, Flow Measurement and Instrumentation 17 (2006) 303–314

[6] Chen FeiXiang, Tian KaiHua, Probe Into Stability Of Higher Capacity Hydraulic Turbine Of Ertan Hydro-Power, Hydropower2004, 68-72

[7] H. Nilsson, and L. Davidson, validations of CFD against detailed velocity and pressure measurements in water turbine runner flow, Int. J. Numer. Meth. Fluids 2003,41:863–879

[8] Sultanian, B.K. Nagao, S.; Sakamoto, T. Experimental and three-dimensional CFD investigation in a gas turbine exhaust system, Journal of Engineering for Gas Turbines and Power, v121, n2, 364-374, 1999

[9] White, J.D. Holloway, A.G.L.; Gerber, A.G. Predicting turbine performance of high specific speed pumps using CFD, Proceedings of the American Society of Mechanical Engineers Fluids Engineering Division Summer Conference, 2005 v2, 125-131

[10] Peng, Guangjie; Qin, Liang; Wang, Zhengwei; Tian, Huijing; Luo, Yongyao, Structure vibration of hydropower house induced by Francis turbine based on CFD, 2008 Proceedings of the ASME Fluids Engineering Division Summer Conference, FEDSM 2008, v1, 1173-1178