Performance study for Francis-99 by using different turbulence models

Zhao Yaping[1]**, Liao Weili, Ruan Hui, Luo Xingqi**
Institute of Water Resources and Hydro-Electric Engineering, Xi'an University of Technology, Xi'an, 710048, China

E-mail: zhaoyp0168@hotmail.com

Abstract. The three-dimensional numerical investigation for turbine-99 at the best efficiency operation point, part load operation point and full load operation point was conducted by using the different turbulence models. By comparing the results of numerical simulation and experimental results, it was found that: there is a certain deviation between the numerical simulation results obtained by different turbulence models and experimental values， and the deviation increase with the reduction of output. Compared to other turbulence model, the result obtained by the standard k-ε turbulent model has a relatively small difference with the experimental results. The main causes for the big difference between the numerical simulation and model test include two aspects: (1) the mesh generation and boundary conditions setting lead to differences between the research object and the actual model, (2) it is difficult to accurately simulate the unstable flow such as impact, flow separation and vortex in the turbine. Therefore, in the future actual flow pattern simulation, besides the reasonable choice of turbulence model, based on the actual flow characteristics, the boundary conditions and the simulation results will be amended to reduce the deviation between the numerical simulation and experimental results as much as possible.

1. Introduction

An increasing number of hydraulic workers focus on accurately simulating the flow characteristics of turbine and estimating its energy characteristics based on CFD numerical computation. Dragica[1] used three turbulent models to predict non- cavitating and cavitating vortex rope in a Francis turbine draft tube, and concluded that SAS-SST, RSM and LES models are suitable for vortex rope prediction and when cavitation was not modeled there was no significant difference in the accuracy of the results obtained by these three turbulent models. S Kurosawa[2] etal simulated the efficiency characteristic of Francis turbine by RSM and LES turbulent models, and obtained that it is necessary to predict the efficiency characteristic by using the unsteady calculation basically. Wu Yulin [3-4] applied SST *k-ω* turbulence model to simulate cavitating flows around a model marine propeller in both uniform flow and wake flow, and proved that SST k-ω turbulence model was more accurate in the simulation of cavitating flow in turbomachineries.

Firstly, several numerical simulations for Francis-99 model were conducted under three operating conditions by means of different turbulence models and boundary conditions. Then, the differences

[1] To whom any correspondence should be addressed.

between the experimental and numerical heads, torques and efficiencies were compared, the uncertainty of turbine efficiency calculation has summarized.

All the model test results for Francis-99 shown in this paper were provided by the web-site of Luleå tekniska universitet - Forskning och utbildning.

2. Numerical simulation method

The computational region of Francis -99 consisted of three components: stationary domain1 (casing, 14 stay vanes, 28 guide vanes); rotating domain (runner with 15blades and 15 splitters); stationary domain-2 (draft tube). The mesh of Francis -99 was provided by workshop, the total number of mesh was 13096727, in which, the stationary domain1 is about 3600000; the rotating domain is about 5170000 and the stationary domain2 is about 3630000. The numerical simulations of Francis turbine at three operating points have achieved, table1 shows the parameters of calculated operating points. Table 2 shows the solution parameters used for performing the numerical simulations.

Table 1. Parameters of calculated operating points

Parameter	Symbol	Part load	BEP	Full load
Net head[m]	H	12.29	11.91	11.84
Flow rate[m^3/s]	Q	0.071	0.203	0.221
Runner angular speed[rev/sec]	n	6.77	5.59	6.16
Guide vane angle[°]	α	3.91	9.84	12.44
Density[kg/m^3]	ρ	999.23	999.19	999.20
Kinematic viscosity[m^2/s]	ν	9.57e-7	9.57e-7	9.57e-7

Table 2. Boundary physics and solution parameters used in the numerical simulations

Parameters	Description
Analysis type	Steady State
Interfaces	Frozen Rotor; discretization type-GGI
Fluid	Water properties updated with actual density, viscosity
Boundary conditons	Inlet: mass flow inlet Outlet: opening-type with static pressure, outlet-type with average static pressure Reference pressure: 0 kPa
Discretization and solution controls	Advection scheme: high resolution Turbulence numeric: high resolution
Turbulence models	Standard *k-ε*, *k-ω* SST
Convergence control	rms of pressure, mass-momentum, and turbulent parameters≤10E-5
Total run	Run-1: three operating points total, standard *k-ε* and opening-type with average static pressure Run-2:three operating points total, *k-ω* SST and opening-type with average static pressure Run-3:three operating points total, standard *k-ε* and outlet-type with average static pressure

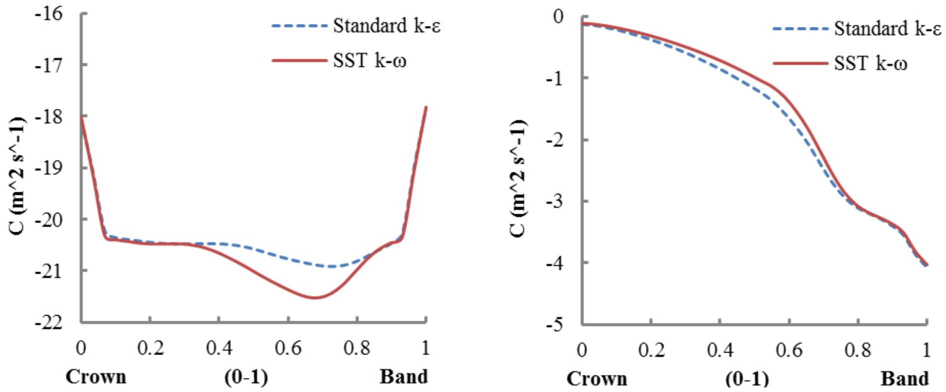
3. Results analysis of numerical simulation

The basic equation of turbine can be described as:

$$H\eta = \omega\left(C_1 - C_2\right)/2\pi g \tag{1}$$

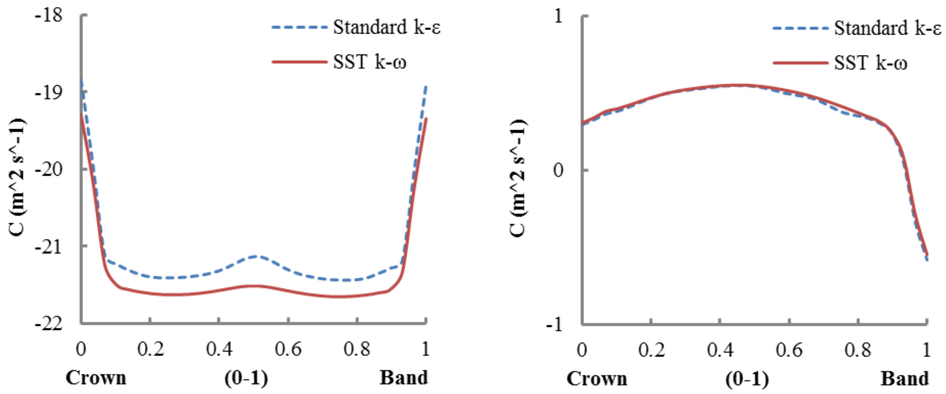
Where, H is the head, η is the hydraulic efficiency, ω is the angular speed, C_1 is the circulation of inlet edge , C_2 is the circulation of outlet edge, g is the acceleration of gravity.

The energy parameters of turbine can be represented by the circulation change amount of inlet and outlet. Figures 1-3 show the average circulation distribution from inlet edge to outlet edge of runner with different turbulent model at three operating points, the circulation difference of runner and rate-of-change of circulation estimated from the standard k-ε turbulence model were smaller than that from the SST model at the same operating conditions.



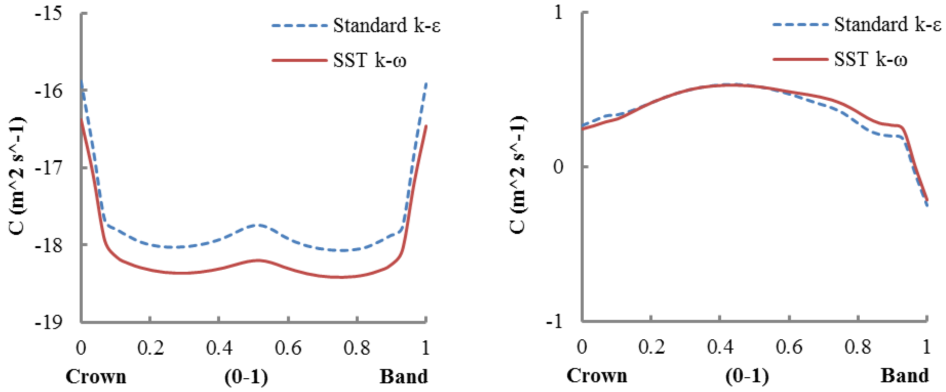
(a) Inlet of runner (b) Outlet of runner

Figure 1. Comparison of the standard k-ε and SST turbulent model circulation at part load operating condition



(a) Inlet of runner (b) Outlet of runner

Figure 2. Comparison of the standard k-ε and SST turbulent model circulation at BEP operating condition



(a) Inlet of runner (b) Outlet of runner

Figure 3. Comparison of the standard k-ε and SST turbulent model circulation at full load operating condition

Figure 4 shows the comparison of torque obtained by the model test and numerical simulation at three operating points. The torque calculated by SST turbulence model and opening-type outlet condition has the maximum deviation compared with experimental torque. The numerical torque is 56.68N m (part load), 81.38 N m (BEP), and 62.22N m (full load) higher than the experimental torque, the ratio with experimental torque was 41.21%, 13.13% and 10.4%, respectively. The numerical torques calculated by standard *k-ε* turbulence model with opening boundary conditions and outlet boundary conditions were similar, slightly less than that computed by SST turbulence model and opening-type outlet condition. Under the condition of standard k-ε turbulence model and opening-type outlet condition, the torque difference between the experimental and numerical was 47.56N m (part load), 71.11 N m (BEP) and 45.032 N m (full load), the ratio with experiment torque was 34.58%, 11.47% and 7.53%.

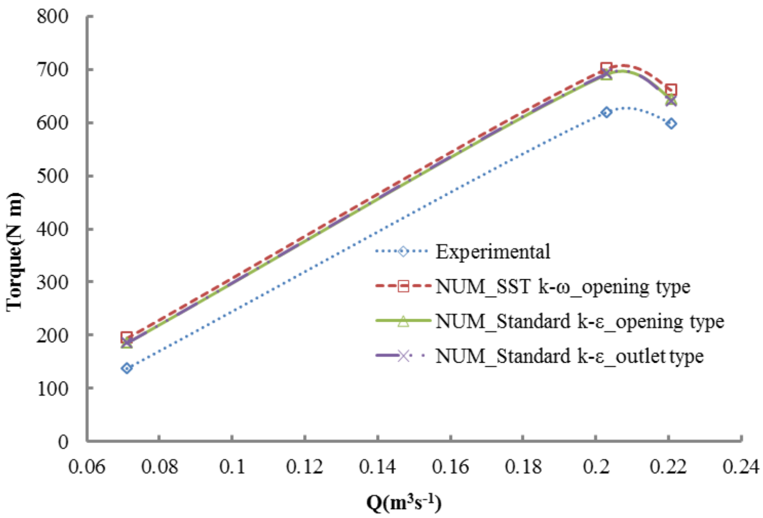


Figure 4. Comparison of the experimental and numerical torque of the turbine at three operating points

Hydraulic efficiency of the turbine can be calculated as:

$$\eta = \frac{T \cdot \omega}{\gamma Q H} \tag{2}$$

Where T is torque, ω is the angular speed, γ is the bulk density of water, Q is the volume flow rate, H is the head.

It can be seen that hydraulic efficiency accuracy of numerical simulation is bound up with the torque, especially in the part load operating conditions, the internal flow of turbine is very complex, unsteady flow such as impact, flow separation and vortex is the main reason caused turbine efficiency reduction.

The comparison of the experimental and numerical hydraulic efficiency at three operating points is shown in Figure 5. Under the opening-type outlet condition, the maximum difference between the experimental and numerical efficiencies was observed at a low discharge (Q=0.071 m^{3}s^{-1}, α=3.91 degree) operating condition. The numerical hydraulic efficiency was 11.06% (k-ε) and 12.42% (SST) higher than the experimental efficiency. The lowest difference between the experimental and numeric results was 0.24% (k-ε) and 1.29% (SST) at the BEP (Q=0.203 m^{3}s^{-1}, α=9.84 degree). The difference between the experimental and numerical efficiencies at the high discharge operating point (Q=0.221m^{3}s^{-1}, α=12.44deg) was 2.71% and 3.67% for k-ε and SST, respectively. The hydraulic efficiency estimated from the standard k-ε turbulence model showed a smaller deviation than that from the SST model.

Under the standard k-ε turbulent model, the simulation with outlet type and the standard k-ε model produced a hydraulic efficiency that was different from the experimental efficiency by 11.17%, 0.74% and 3.06% for low discharge, BEP and high discharge respectively. The hydraulic efficiency estimated from the outlet type model showed a higher deviation than that from the opening type model.

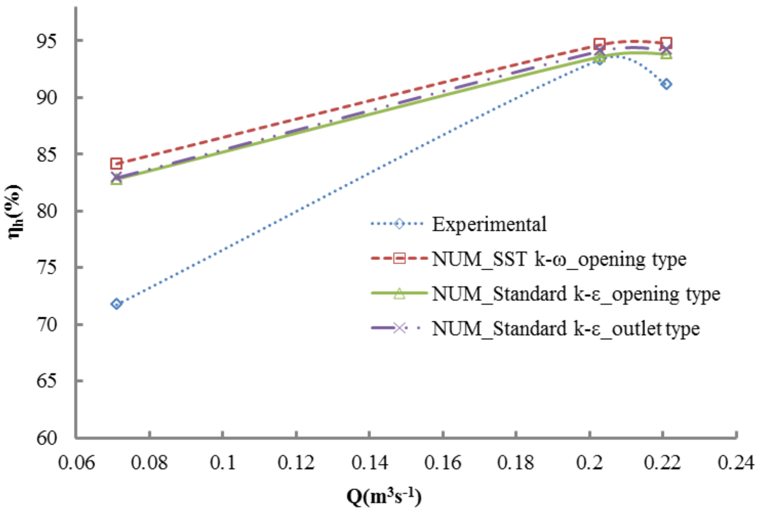


Figure 5. Comparison of the experimental and numerical hydraulic efficiency of the turbine at three operating points

Figure 6 shows the head comparison of the experiment and numerical simulation. The maximum difference occurs at the low discharge operating conditions, the difference between the experimental and numerical head was 1.35m (k-ε, outlet), 1.37m (k-ε, opening) and 1.81m (SST, opening). The ratio of head difference and experimental head was 11%, 11.14% and 14.72%, respectively. At BEP operating point, the numerical head was 1.1m (k-ε, outlet), 1.13m (k-ε, opening) and 1.19m (SST, opening) higher than the experimental head, the ratio of head error and experimental head were 9%, 9.48% and 9.99%. The smallest head difference appears in the high discharge operating point, the difference between numerical simulation and experimental head was 0.29m (k-ε, outlet), 0.41m (k-ε, opening), 0.61m (SST, opening), the ratio of head difference and experimental head were 2%, 3.46% and 5.15% ,respectively. It can be seen that head difference of the standard k-ε turbulence model and outlet boundary conditions simulation was the least relatively.

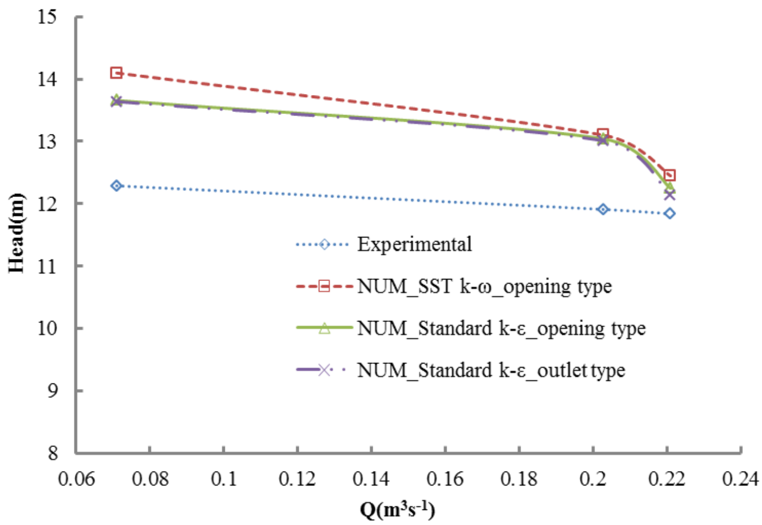


Figure 6. Comparison of the experimental and numerical head of the turbine at three operating points

4. Conclusions

The numerical simulations with the standard k-ε turbulence model, the SST turbulence model, the opening and outlet boundary conditions respectively have been fulfilled, and the results of numerical simulation are compared with experimental data, the following conclusions are obtained:

1) The larger circulation deviation of SST turbulence model simulation lead to the higher hydraulic efficiency than the standard k-ε turbulence model simulation. And the differences between the experimental and numerical efficiency, head and torque simulated by standard k-ε turbulence model are smaller than that simulated by SST turbulence model.

2) The minimum difference operating point between the experimental and numerical efficiencies was the BEP operating condition, high discharge operating point seconded, and low discharge operating condition is the largest.

3) The unstable flow such as impact, flow separation and vortex may cause the torque calculation inaccuracy in the numerical simulation, which lead to the numerical efficiency greater than the experimental data.

So, besides the reasonable choice of turbulence model, how to accurately amending the boundary conditions and the simulation results will be an urgent problem to reduce the deviation between the numerical simulation and experimental results.

References:

[1] Dragica Jost-Andrej Lipej. Numerical prediction of non-cavitating and cavitating vortex rope in a Francis turbine draft tube. Journal of Mechanical Engineering 57(2011)6, 445-456.
[2] S Kurosawa, S M Lim and Y Enomoto. Virtual model test for a Francis turbine. IOP Conf. Series: Earth and Environmental Science 12(2010) 012063.
[3] B.Ji,X.W.Luo,X.X.Peng,Y.L.Wu and H.Y.Xu. Unsteady numerical simulation of cavitating turbulent flow around a highly skewed model marine propeller, Journal of Fluid Engineering, 133(1)(2011)011102-1-8.
[4] Yulin Wu, Jintao Liu, Yuekun Sun, Shuhong Liu and Zhigang Zuo. Numerical analysis of flow in a Francis turbine on an equal critical cavitation coefficient line. Journal of Mechanical Science and Technology 27(6)(2013)1635-1641.