Numerical investigation of cavitation performance on bulb tubular turbine

L G Sun¹, P C Guo¹, X B Zheng¹ and X Q Luo¹

¹ State Key Laboratory Base of Eco-hydraulic Engineering in Arid Area, Xi’an University of Technology, Xi’an 710048, Shaanxi Province, China

E-mail: 15191587315@126.com

Abstract. The cavitation flow phenomena may occur in the bulb tubular turbine at some certain operation conditions, which even decrease the performance of units and causes insatiably noise and vibration when it goes worse. A steady cavitating flow numerical simulations study is carried out on the bulb tubular unit with the same blade pitch angle and different guide vane openings by using the commercial code ANSYS CFX in this paper. The phenomena of cavitation induction areas and development process are obtained and draws cavitation performance curves. The numerical results show that the travelling bubble cavity is the main types of cavitation development over a wide operating range of discharge and this type of cavitation begins to sensitive to the value of cavitation number when the discharge exceeding a certain valve, in this condition, it can lead to a severe free bubble formation with the gradually decrement of cavitation number. The reported cavitation performance curves results indicate that the flow blockage incident would happen because of a mount of free bubble formation in the flow passage when the cavity developed to certain extend, which caused head drop behavior and power broken dramatically and influenced the output power.

1. Introduction

Bulb tubular hydro power station is one of the best ways to develop low water head hydropower resources due to some distinguished features of bulb tubular turbine such as excellent technical and economical characteristics and widely applicability, it has become a hot issue and focus for the bulb tubular turbine in the field of development, design and operation technology and experience. Furthermore, the whole flow passage of the power station is basically in the axial direction with less hydraulic loss and convenient for construction, and strong discharge capacity higher specific speed are another particular advantage on the basis of equal water head and power, which also result in a great decrease of the units[1]. Cavitation may be observed in various engineering systems and is usually the main physical phenomenon behind performance alteration in hydraulic machinery, and modern design requirements lead to more compact machines with higher rotation speeds and cavitation risk. However, it’s a great difficult and changing task to predict such complex unsteady and two-phase flows as well as its inception and development with an acceptable accuracy [2].

The paper [3] presents the cavitation phenomenon characterized by fluid machinery including type of cavity development related to the specific speed of machines in both pump and turbine mode, the influence of the operation conditions, such as load, head and submergence, also, the influence of cavitation development on the machine efficiency, operation and integrity are discussed. In paper [4], authors has put forward a new “full cavitation model” for performance predictions of engineering
equipment under cavitation flow conditions with consideration of effects of turbulent fluctuations and non-condensable gases. In addition, final validation results towards this new cavitation model are presented for flows over hydrofoils, submerged cylindrical bodies, and sharp-edged orifices. Another application of the full cavitation model to pumps and inducers is given for instance by paper [5] by means of simulations of cavitating flows in three types of machines: water jet propulsion axial pump, a centrifugal water pump, and an inducer from a LOX turbo pump. The results show cavitation zones on the leading edge-suction side of each of the machines as expected and all the test cases with cavitation show plausible results. In paper [6], unsteady cavitation turbulent flow around the 3D twisted Delft hydrofoil was simulated by using the Partially-Average Navier-Stokes(PANS) method and a mass transfer cavitation model with the maximum density ratio effect between the liquid and the vapor, the numerical results indicate that the cavity volume fluctuates dramatically as the cavitating flow develops with cavity growth, destabilization and collapse, which presents good agreement with the existing experiment observations. The paper [7] deals with the basic investigation towards the time-dependent cavitating flow around an inclined 2D profile to develop the suitable bubble dynamic model and describes the application of this model to the prediction of the time-average leading edge cavitation and the head drop behavior of a centrifugal pump. Paper [8] employed numerically compare case of a bulb tubular turbine with blade tip gap, with hub gaps and without any gap and presents the impacts of above three types gaps on performance predictions, furthermore, the RANS simulations of the efficiency hill chart of a bulb turbine was investigated. In paper [9], a flow in a horizontal shaft bulb turbine is calculated as a two-phase flow with a commercial Computational Fluid Dynamics code including cavitation model. The results are compared with experimental results achieved at a closed loop test rig for model turbines.

This paper presents steady numerical simulations of cavitating flow study on the model bulb tubular unit with the same blade pitch angle and different guide vane openings by applying the Zwart cavitation model derived from a simplified Rayleigh-Plesset equation using the commercial code ANSYS CFX with the SST turbulence model. The phenomena of cavitation induction areas and development process are obtained and drawn cavitation performance.

2. Calculation model and grid generation

2.1. Mode and computation conditions

Hydraulic losses derived from design of flow passage constitute the majority of weight due to a low water head and large discharge for the bulb tubular turbine. It necessitates a reduction in hydraulic loss with reasonable designing the shape and size of the flow passage, as well as low project cost. In this paper the basic parameters of the investigated model turbine is reported in table 1. The schematic diagram of the flow passage corresponding to table 1 is showed in figure 1, and figure 2 presents the three-dimensional model for the simulation domain, which is the application of the three-dimensional modeling software to generate the calculation domain. Figure 2 shows that the unit is composed of 5 parts, which are respectively the diversion chamber, the bulb body, the guide vane, the runner and the draft tube.

| Parameters                   | Remark | Parameters                   | Remark |
|------------------------------|--------|------------------------------|--------|
| Runner diameter(D₁)         | 350mm  | Hub diameter(dₕ)             | 140mm  |
| Text head(He)                | 4m     | Blade numbers(Z₁)            | 4      |
| Guide Vane numbers(Z₂)      | 16     | Guide vane height(b)         | 120.7mm|
According to the cavitation flow characteristics of tubular turbine, it can be known that the cavitation is more susceptible to be triggered at higher mass flow rates, which is one key factor to select operation conditions, the another is the method of controlling vitiating. In order to comparatively simulate different trends of cavitating flows in the model turbine, two on-cam operation points that is featured by the same blade pitch angle and different guide vane openings are selected to carried out the numerical calculation, corresponding to points A and B in figure 3. The figure indicates that those two operating conditions share the same blade pitch angle of 25 degree, while respectively 90% and 102.5% of the guide vane openings.
2.2. Grid generation
One of the most time-consuming tasks in CFD is the generation of grid in the computation domain, particularly for hydraulic machinery, the flow passage components is not only a complex space surface, but also the periodic and non-periodic distribution of complex surfaces. The computation model is discretized into multi-block structured grids based on the grid generation tool ICEM CFD. Figure 4 shows the grid generation of the investigated turbine, an H&L topology is employed in the blade flow passage and an O-grid is created around the blade, which will provide opportunities of excellent boundary layer resolutions. The grids are well refined at the zones around the walls in order to more accurately simulate the viscous flow near the solid wall.

![Grid generation of the investigated turbine](image)

(a) Diversion chamber and bulb body  (b) Guide vane  
(c) Runner  (d) Draft tube  

**Figure 4.** Grid generation of the investigated turbine

3. Numerical computational conditions
The finite volume solver ANSYS-CFX is utilized to govern the Reynolds-Average Navier-Stokes equations with SST turbulence model. The inlet of the calculation domain is specified by using the averaged mass flow-rate based on the mass equilibrium. At the outlet of draft tube, an averaged static pressure is given according to the total pressure level. All solid walls are non-slip and smooth. Defining the interfaces between rotating parts and stationary parts as "Frozen Rotor" slip interfaces at steady-state computation, which treats the flow from one component to the next by changing the frame of reference while maintaining the relative position of the components and has the advantages of being robust, using less computer resources.

In addition, different cavitation development trends in runner already mentioned are observed by means of changing the absolute pressure of draft tube outlet, the “Schnerr-Sauer” cavitation model based on homogeneous multi-phase transport equation is employed for cavitation simulation with vaporization pressure 3540 Pa at 25 °C, other empirical constants are taken as proposed by ANSYS.
4. Cavitation flow results analysis

Design and operation of hydraulic turbines are strongly related to cavitating flow phenomenon, which may appear in either the rotating runner or the stationary parts. The cavitation development may be the origin of several negative effects, such as noise, vibrations, performance alterations, erosion and structural damages. The cavitation flow numerical results of vapor volume fraction on runner blades and cavitation performance curves are respectively reported from figure 5 to figure 10 at two operation conditions A and B. In figure 5, the cavitation zones are enlarged for better views, and the inlet edge of runner blade on the left in figure 6, figure 7 and figure 8, $\sigma$ is cavitation number.

Figure 5. Vapor volume fraction on pressure side at operation condition A with different $\sigma$
Figure 6. Vapor volume fraction on suction side at operation condition A with different $\sigma$

(a) $\sigma=2.55$  (b) $\sigma=2.04$  (c) $\sigma=1.51$

(d) $\sigma=1.25$  (e) $\sigma=0.98$  (f) $\sigma=0.86$

(g) $\sigma=0.78$  (h) $\sigma=0.72$

Figure 7. Cavitation performance curves at operation condition A

(a) Head drop curves  (b) Output broken curve
Figure 8. Vapor volume fraction on pressure side at operation condition B with different $\sigma$

(a) $\sigma=2.66$  (b) $\sigma=2.40$  (c) $\sigma=2.11$

(d) $\sigma=2.04$  (e) $\sigma=1.97$  (f) $\sigma=1.89$

Figure 9. Vapor volume fraction on suction side at operation condition B with different $\sigma$

(a) $\sigma=2.66$  (b) $\sigma=2.40$  (c) $\sigma=2.11$

(d) $\sigma=2.04$  (e) $\sigma=1.97$  (f) $\sigma=1.89$
In generally, the majority of vapor can be observed on the suction side of the runner blades although is also occur on the pressure sides at operation condition A, especially towards high $\sigma$. It can only observe slight trend cavitation on pressure side due to small negative angle of attack is generated at condition A, while greater negative angle of attack would appear if the flow rate is much higher than the design valve, which results in flow separation and has an obvious influence on cavitation induction. Therefore, the vapor volume fraction on pressure sides at B condition is greater than A. The volume of the regions affected by cavitation increases with decreasing cavitation number and increasing flow rates on suction sides a, while it has no significant change on pressure side.

The suction side is the main zones that cavitation occurs and a cavity development takes place at the leading edge of the runner blades firstly, then hub and in the vicinity of the shroud along with the direction of chord length at condition A, and eventually occupied the outlet edge of the runner blade. These leading edge cavities can be avoided by improving the hydraulic design of runner. For operation condition B, the development process of cavitation on the blades suction side can be concluded, the traveling bubble cavity development takes place at the hub of the runner and the cavitation cloud will be more serious at lower outlet pressure. An isolated and banded cavity cloud was observed at the blades tip, and the fully developed cavitation area would connect with the hub cavitation region when the cavitation number declined.

Usually some amount of cavitation is allowed during normal operation for the bulb tubular turbine as the cavitation performance shows in figure 8 and figure 10, which also indicate the influence of cavitation trends to external characteristics of the turbine. The water and output remains relatively constant when $\sigma$ variates in a certain range, while cavitation performance is very sensitive to $\sigma$, which results in a sharp decrease if $\sigma$ exceeds a certain value. Furthermore, water and output has seriously reduced in spite of light cavitation occuring in suction sides for higher mass flow rates by comparision analysis between figure 9 and figure 10 due to flow blockage incident derived from amount of free bubble formation in the flow passage.

5. Conclusion

In this paper, the three-dimensional numerical calculation of cavitation flow for a model bulb tubular turbine is carried out, the cavitation induction areas and development process are obtained and draws cavitation performance curves. The numerical results show that the travelling bubble cavity is the main types of cavitation development on the suction sides of runner blades and this type of cavitation begins to sensitive to the value of cavitation number when the discharge exceeding a certain valve, which leads to a sharp reduction in performance.
6. Reference

[1] Tian S T 2003 *Tubular turbine generator set and its selection method* (Beijing: China Electric Power Press)

[2] Pan Z Y and Yuan S Q 2013 *Fundamentals of cavitation in pumps* (Zhenjiang: Jiangsu University Press)

[3] Avellan F 2004 Introduction to cavitation in hydraulic machinery The 6th International Con. on Hydraulic Machinery and Hydrodynamics (Timisoara, Romania 21-22 October)

[4] Singhal A K, Athavale M M, Li H Y and Yu J 2002 Mathematical basis and validation of full cavitation model J. *Journal of Fluid Engineering*. 124 617-24

[5] Athavale M M, Li H Y, Yu J and Singhal A K 2002 Application of the full cavitation model to pumps and inducers *J International Journal of Rotating Machinery*. 8(1) 45-56

[6] Ji B, Luo X W, Wu Y L, Peng X X and Duan Y L 2013 Numerical analysis of unsteady activating turbulent flow and shedding horse-shoe vortex structure around a twisted hydrofoil *J International Journal of Multiphase Flow*. (51) 33-43

[7] Frobenius M and Schiling R 2004 Cavitation prediction in hydraulic machinery 22nd *IAHR Symp. on Hydraulic Machinery and Systems*. (Stockholm, Sweden June 29-July 2)

[8] Guénette V, Houde S, Ciocan G D, Dumas G, Huang J and Deschênes C 2012 Numerical prediction of a bulb turbine performance hill chart through RANS simulations 26nd IAHR Symp. on Hydraulic Machinery and Systems. (Beijing, China 19-23 August)

[9] Necker J and Aschenbrenner T 2010 Model test and CFD calculation of a cavitating bulb turbine. *25th IAHR Symp. on Hydraulic Machinery and Systems*. (Timișoara, Romania 20-24 September)