Research on the flow field of variable head water flow standard facility based on numerical simulation method

Haiyang Li ¹, Liang Zhang ²,*, Bian Hong ¹, Xuejing Li ¹ and Yuyang Chen ¹

¹ Shanghai Institute of Measurement and Testing Technology, Shanghai, China.
² National Institute of Metrology, China, Beijing, China.

* zhangliang@nim.ac.cn

Abstract. This paper introduces a basic method of using numerical simulation to calculate the internal flow field of variable head water flow standard facility. The content includes the procedure of 3D geometric modeling, mesh gridding and boundary condition setting. The numerical results visually shows the flow filed in the facility, of which proved the correctness and feasibility of the method. The simulation results provide constructive suggestions for the design and manufacture of variable head water flow standard facility.

Keywords: variable head water flow; numerical simulation method; flow field.

1. Introduction

The large diameter liquid flowmeters are widely used in the measurement of production water, urban domestic water. The common liquid flowmeters are generally calibrated by static mass method, volumetric method, and standard meter method. However, there are some limitations in the calibration of large diameter flowmeters using these methods. As a stable source of water flow, high-level constant pressure water tower consumes a lot of energy, and the parallel use of multiple standard meters will introduce more sources of uncertainty. While, the dynamic volumetric water flow standard facility, based on the principle of variable head, overcomes the shortcomings of the above standard facility, and has obvious advantages in reducing energy consumption and effective utilization of water resources [1,2].

However, due to the lack of continuous power support for the variable head water tower, the following problems should be considered in the design of the facility: whether there is vortex inside the water tower [3], whether the liquid level is stable, where is the effective utilization height of the water tower, and where are the appropriate positions for the installation of the level sensors. Because of no appropriate methods of detection means exist, the solution of the above problems mainly depends on the numerical simulation method.
2. Introduction of variable head water flow standard facility

2.1. Facility composition
The composition of variable head water flow standard facility is shown in Fig. 1, which mainly includes the water tower, the level sensors, the temperature sensor, the pressure sensor, the rectifier, various types of valves, the pool, the converter, and the computer.

![Fig. 1 The composition of variable head water flow standard facility](image)

2.2. Working principle.
The volume between two liquid level sensors in the water tower is called a standard volume section, and the water tower is composed of several standard volume sections which volumes have been calibrated in advance. When the flow standard facility works, with the water level in the water tower drops, the flow rate will also decrease. In order to keep the flow rate basically constant, the opening of the flow regulating valve should be increased continuously. The timer measures the time interval between the two level sensors when the water drops in the water tower. The relative error of the tested flowmeters can be obtained by comparing the accumulated flow of the tested flowmeters with the standard volume given by the standard facility.

3. Numerical simulation method of variable head water flow standard facility
Since the 1970s, the computational fluid dynamics (CFD) has developed rapidly. Experimental research, theoretical analysis and numerical simulation have become three basic methods to study the law of fluid motion. Based on the actual variable head water flow standard facility of a flow meter manufacturer in Shanghai, this paper introduces the method of numerical simulation to study the internal flow field of water tower and its horizontal outlet pipeline.

3.1. Geometric model and mesh generation
The geometric dimensions of the flow standard facility studied in this paper are as follows: the bottom inner diameter of the variable head water tower is 4400 mm; the top inner diameter is 5800 mm; the height of the water tower is 27500 mm (from the second level sensor, where the liquid level of the first-2 sections are left blank); there is a diameter-change at 1500mm of the water tower; the diameter of the outlet pipeline is 3000mm; and the length of the pipeline is 63180mm. The flow range of the standard facility is 600m$^3$/h-25000m$^3$/h. The modeling software was used to reconstruct the geometry of the facility and the calculation domain included variable head water tower and horizontal outlet pipeline. The 3D geometric model of the facility is shown in Fig. 2.
Then the above 3D geometric model was meshed. Considering the quality of mesh and the accuracy and speed of numerical calculation, structured hexahedral mesh was adopted, and the mesh is shown in Fig. 3.

3.2. Turbulence model and boundary condition setting

As the flow in the water tower is unsteady, the pressure-transient simulation method was adopted in the solver. When the water flow standard device works, the air at the top enters into the water tower with the water level falling down. There is an obvious gas-liquid interface. So the computational domain contained 2 mediums of gas and liquid which belonged to the category of multiphase-flow. Therefore, the VOF model of multi-phase flow was used to capture the water-air interface, and the interfacial tension was set as 0.072N/m. The inlet boundary of calculation domain was the top of water tower and was set as pressure inlet; the outlet boundary was the end of the horizontal outlet pipeline and was set as mass flow outlet. Other tower walls and pipe walls were set as stationary wall. The solid-liquid contact angle was set as 45°, and the time step was 0.03s. In order to reduce the consumption of computing resources, the RANS (Reynolds-averaged Navier-Stokes equations) model was chosen as the turbulence model on the premise of ensuring the calculation accuracy.

4. Numerical calculation results of variable head flow standard facility

This part presented the water level falling curve of the whole water tower, the overall streamline diagram, the velocity contours and velocity vector diagrams of the different positions of the water tower and the outlet pipe, so as to study the internal vortex and the flow field of the water tower and the outlet pipeline, the effective height and “Dead-area” of the water tower[4,5].
4.1. Analysis of water level falling process of the variable head water tower
In this paper, the average height of water level (the average height of 46 detection points in the radial direction of a water level in the tower) in the tower at different times is observed under the condition of given outlet flow of 25000 m³/h to study the fluctuation of the water level and the severity of water level change at wall points in the process of the water level falling. The position of the 46 measuring points is shown in the figure 4.

![Fig. 4 The position of the 46 measuring points](image)

Based on the height of the detection points in the above figure, the following average height-time and water level change rate - height curves were drawn. Figure 5 showed the curve of the water level with time change of average height of 46 radial points and figure 6 showed the curve of their’s level falling rate with water height change.

![Fig. 5 The curve of the water level with time change of the 46 radial points](image)

![Fig. 6 The curve of water level falling rate with water height change of the 46 radial points](image)
From figure 5 and figure 6, it can be concluded that the descent rate of water level at gas-liquid interface (at the position above the diameter change height) was about 0.24 m/s, and the fluctuation range was about 8% with the rate changing all the time. And after the diameter change the descent rate increased and finally stabilized, at about 0.44 m/s, and the fluctuation range of descent rate was not more than 4%. That is, the water level in the water falling process in the fine water tower was more stable than that in the coarse water tower. Analyzing the reasons for the above situation, on the one hand, it was related to the existence of the diameter change of the water tower, which hindered the continuity of the flow field. The flow velocity before the diameter change was small, and the relative fluctuation percentage caused by the same absolute velocity fluctuation was larger. Between the diameter change section, vortices with different scales were produced, which affected the stability of the descending velocity of the gas-liquid interface. On the other hand, there was a strong velocity gradient near the outlet of the water tower, which had a certain impact on the limited height of the water level in the tower. Because there were many different scale vortices on the single water level, it was still seemed relatively stable on the whole of a single level.

4.2. Analysis of flow field in the water tower at a certain time
This section studied the typical flow field of the water tower at a certain time. The specific shape of the flow field were analyzed by selecting the velocity vector diagram, streamline diagram, pressure and velocity contours of the facility at the time of 18.0s and still under the condition of outlet flow rate of 25000 m$^3$/h.

The above figure shows the streamline diagram of inner water tower at the time of 18.0s. It can be seen that there was basically no streamline passing through the lower left corner of the tower, and the streamline bends upward to the right. The water skipped over the dead zone at the bottom of the water tower, where there was a strong vortex. In addition, from the inlet to the outlet of the tower, the streamline shape was relatively straight, indicating that the liquid surface is relatively stable, and there is no large vortex on the whole.
Fig. 8 The velocity vector of the central cross section and local Enlargement at the time of 18.0s

The above two figures show the velocity vector diagram of the central cross section of the facility at 18.0s. It can be seen from these figures that the area with the largest velocity gradient in the whole flow field appears at the outlet of the water tower, and the affected range of the outlet pipeline extended for more than 16.0m which was near to the end of the pipeline. And, it also can be seen that the velocity distribution changed greatly and the ideal velocity profile pulses obviously in the direction of gravity due to the influence of large eddy flow at the outlet and the variable diameter section in the middle of the tower.

Fig. 9 The velocity vector diagram of different cross section
The figure 9 shows the velocity vector diagram of different cross section of the horizontal outlet pipeline at 18.0s. It can be seen from the figure that the position of the largest velocity gradient was closest to the outlet of the water tower (x = 2.2m) among all the selected positions and the maximum velocity was about 1.5m/s, which was mainly distributed the upper part of the outlet pipeline. Two strong vortex regions were distributed at the left and right symmetrical positions of the cross-section respectively. The velocity distribution of the cross-section in the outlet pipeline was extremely uneven. The four vortices developed from X = 2.2m to X = 8.2m, reaching the peak of their influenced area. After that, the vortex was restrained by more and more uniform water flow, and its intensity decreased continuously. Until X=16.2m, the velocity of the cross-section was nearly uniform, and no obvious vortex was exist. Finally, the flow field was fully developed and suitable for flowmeter calibration. From the figures of X=8.2m and X=12.2m, the four vortices influenced the flow field of the pipeline for 50%. The obvious velocity fluctuation directly affected the flow field inside the tower, thus the large-scale spatial inhomogeneity along the Y-axis of gravity was observed at the gas-liquid interface and below the interface, and the lateral pulsation was small. Therefore, it is suggested that the uniform flow field should be selected when calibrate the flowmeters according to the numerical simulation results to reduce the uncertainty of measurement results caused by eddy flow or disturbance.

5. Summary
This paper introduces a basic method of using numerical simulation to simulate the internal flow field of variable head water flow standard facility. From the establishment of geometric model, to the generation of mesh and finally to acquisition the flow field of the standard facility, the correctness and feasibility of the above method are proved. The numerical results visually showed the flow stability in the process of water level falling in the water tower, dead zone at the bottom of the tower, and existence of sever vortex in the front of the horizontal pipeline. The appropriate position of the flowmeters to be calibrated installed on the horizontal pipeline is recommended. In summary, the simulation results provide constructive suggestions for the design and manufacture of variable head water flow standard facility, of which have practical guiding significance.

Acknowledgments
This work was financially supported by the National Science and Technology Support Program Foundation (Grant NO. 2017YFF0205301 and Grant NO. 2018YFF0212405).

References
[1] Yu Kang, Cheng Yongguang, ZHANG, Xiaoxi. Hydraulic characteristics of a siphon-shaped overflow tower in a long water conveyance system: CFD simulation and analysis. Journal of Hydrodynamics. Vol. 28(2016), p. 564-575.
[2] Li, J.Y., Wang, H., Sheng, W. Numerical simulations for super large-scale natural draft cooling tower with an optimized non-uniform water distribution. 25th International Conference on Nuclear Engineering. Proceedings, edited by American Society of Mechanical Engineers, ShangHai, 2017.
[3] SARKARDEHH. Numerical calculation of air entrainment rates due to intake vortices. Meccanica. Vol.52 (2017), p. 3629-3643.
[4] Li, H.F., Chen, H. X., Ma, Z., et al. Experimental and numerical investigation of free surface vortex. Journal of Hydrodynamics. Vol. 20 (2008), p. 485-491.
[5] Zhang C., Sun X., Li Y., Zhang X., et al. Effect of fluid-structure interaction on internal flow field characteristics of tube-contained raw material pipeline hydraulic transportation. Nongye Gongcheng Xuebao. Vol.34 (2018), p. 299-307.