Optimization of Land-Saving Underground Pump Station Combining Computational Fluid Dynamics and Physical Model Experiment

Shuo Zhang*, Ruhua Wang
Shanghai Municipal Engineering Design Institute (Group) Co., Ltd., Shanghai, 200092, P.R. China
*Corresponding author’s e-mail: zhangshuo@smedi.com

Abstract. Combining 3D CFD numerical simulation and physical model experiment is applied in underground pump station for saving land and energy. The pump station's overall structure is targeted at a large analysis area. Hydraulic rectifications such as diversion plate and distribution hole are confirmed for better hydraulic conditions and distributing uniformity evidently. Based on comparing project schemes and optimal design, land is approximately saved half and head loss is reduced about fifty eight percent.

1. Introduction
Large-scale water pumping stations have more investment, and stable and reliable operation plays an important role in water supply safety. Selection range of design parameters is large in the code and standards. Therefore, direct design of large pumping stations based on design and engineering experience has certain risks and may not be economical [1].

A large pumping station with long distance water diversion not only is large in scale, but also has a large number of pumps and a high lift and a large energy consumption [2]. And there are several construction conditions with many challenging designs [3]. Thus, various schemes were compared and optimized with CFD numerical simulation and physical model experiment in order to effectively protect the scientific design and safe operation.

2. Experiment method

2.1. CFD
The physical geometry of the relevant calculation area such as the pump station inlet pool is generated via Pro/E software. Geometry files using ICEM grid generation software are imported with geometric inspection and repair. The block hexahedral body mesh is generating by a combination of topologies such as O, C, H and J [4]. The retractable wall function method is used to treat the near wall area [5]. For the free surface, the method of setting the symmetry plane is adopted. CFD software is Ansys Fluent.

2.2. Physical Model
A physical model by gravity similarity criteria is built as similar ratio eight [6]. Water flow velocity in three-dimensional flow field is measured by point flow meter Vectrino.
3. Results and discussion

3.1. Primary election with CFD
Four preliminary schemes for selecting long strip, figure eight, sector and rectangle shapes were compared by CFD within short time and credible accuracy (figure 1,2,3,4). The trapezoidal front pool symmetrical scheme was initially recommended. After statistics, head loss was mainly concentrated in two transitions. The hydraulic loss generated in other parts was relatively small (table 1).

![Long strip shape](image1.png)
![Figure eight shape](image2.png)
![Sector shape](image3.png)
![Rectangle shape](image4.png)

Table 1. The recommended maximum water level drop in the model (m)

|                          | Inlet well | Head of distribution channel | End of distribution channel | End of front pool | Total head loss |
|--------------------------|------------|------------------------------|-----------------------------|------------------|-----------------|
| Maximum daily            | -4.287     | -4.431                       | -4.451                      | -4.479           | 0.192           |
| Average daily            | -2.584     | -2.690                       | -2.704                      | -2.724           | 0.140           |
| Regular salt-resistant   | 1.806      | 1.738                        | 1.734                       | 1.718            | 0.088           |
| Occasional salt-resistant| -5.894     | -6.016                       | -6.036                      | -6.056           | 0.162           |
| Accident                 | -7.058     | -7.196                       | -7.212                      | -7.224           | 0.166           |

3.2. Combining CFD and Physical Model
Combined with the results of the physical model test, three-dimensional geometric models of four different schemes were established. CFD flow patterns under design conditions were analysed and compared with the results of the physical model test.
3.2.1. Scheme One. The program was recommended preliminarily by CFD. The flow result calculated by numerical simulation was close with the result of physical model test observation. The vortex of the front pool was obvious and flow was unsteady (figure 5,6).

3.2.2. Scheme Two. On the basis of Scheme One, two holes walls were added in the inlet well. Short water guiding walls and eight-shaped diversion piers were respectively added in the water distribution channel and the front pool. For the most distal front pool, the flow state obtained by numerical calculation was basically the same as that of the physical model test. The uniformity of lateral distribution of flow velocity in each inlet flow channel has been improved and reverse flow was eliminated. The flow pattern became relatively stable (figure 7,8). But head loss increased to 0.326m because of rectification.

3.2.3. Scheme Three. On the basis of the Scheme Two, a hole wall was reduced and a sill was added in the inlet well. The flow pattern in the inlet, distribution, front and inlet channels was improved and head loss reduced to 0.326m. After the addition of the bottom ridge, the flow pattern of the inlet well changed then turbulence was reduced. The drift of the distribution channel had improved with more uniform distribution of water flow into the front pool. The vortex intensity in the front pool was reduced. The flow direction was reasonable on different sections of the inlet runner (figure 9,10). Flow rate at the bottom was large and lateral uniformity was better.
3.2.4. Scheme Four. On the basis of the Scheme Two, the hole wall was canceled in the inlet well. Slope transition was set between the sill of the inlet well and the distribution channel. Another sill was added in the bottom of the front pool close to the side of the distribution channel. The flow pattern was more stable and head loss reduced to 0.192m which was 58% less than empirical parameter design (figure 11,12). The flow pattern of the inlet well became better then turbulence was lower because of the sill. The distribution of water flow into the front pool were more uniform. Scheme Four as the final practical solution was applied to the station. Thus, land was approximately saved half and head loss was reduced about fifty eight percent. At present, the pumping station operates in good condition.

3.2.5. Water flow regime under maximum daily condition. The water flow in the distribution channel was smooth except a vortex at the end. The distribution of water flow in the distribution channel into the front pool was uniform (figure 13,14,15,16,17,18). The water flow in the front pool will be biased because the water flowed through the 90° turn into the front pool. The velocity of the outer boundary of the vortex was about 0.05 m/s showing weak vortex strength.
3.2.6. Water flow regime under average daily condition. There was a pump that didn't work under Z=2.5m. Due to less flow, the flow in the front pool and the distribution channel was smoother and the vortex strength in the front pool was weaker. The flow rate was gradually homogenized from the inlet in the flow direction especially good lateral uniformity.

3.2.7. Water flow regime under accident condition. An inlet pipe did not work under another inlet pipe operation. The flow velocity at the bottom of the distribution channel was significantly larger than the flow velocity at the upper and water depth was small. Hence there were strong vortex and recirculation in the water distribution zone. The position of the vortex was in the rear area of the front pool area. Since the water inlet of the connecting water distribution channel and the front pool were at the upper part, the mainstream was at the upper end. After the water flows into the front pool, there was a case where the upper flow velocity was large and the bottom flow velocity was small which continued until
the inlet channel. With the flow process, the flow velocity distribution of the water flow was evenly homogenized. Therefore, the water flow in the inlet flow channel was smooth, and the flow condition in the lower part of the bell mouth was good (figure 19, 20, 21, 22, 23, 24).

4. Conclusions
The numerical simulation of design schemes by CFD could recommend solutions for physical model testing and design. The CFD would analyse the location where the head loss is concentrated and show the improved hydraulic conditions and the reduced head loss under rectification measures.

Digital model results and engineering measures were tested by the flow detection in the physical model experiment based on the recommended solution for numerical simulation results. Several
optimization methods for adding eight-character diversion piers and bottom sills were proposed and could achieve stable water flow, even water distribution and reduce head loss.

Under the premise of ensuring hydraulic stability and uniform water distribution, land was approximately saved half and head loss was reduced about fifty eight percent based on comparing project schemes and optimal design. The design goal of system reliability and energy saving was achieved.

Acknowledgments
The authors would like to acknowledge the financial support by the key Special Program on the S&T for the Pollution Control and Treatment of Water Bodies (No.2017ZX07501001-05) and the Shanghai Science and Technology Committee Program (No.16DZ0503500).

References
[1] Feng, J.G., Wang, X.S., Tong, H.W. (2010) Improving of the flow patterns in forebays of the large urban water supply pumping stations. Water & Wastewater Engineering., 36: 51-54.
[2] Wang, R.H., Shen, P.Y., Li, J.Y. (2009) Research on the pump style selection of exceptionally large and deep pumping station. Water & Wastewater Engineering., 35: 50-55.
[3] Li, J.Y., Li, G.P., Wang, R.H. (2011) Water level variation analysis of pre-tank during the No.5 ditch pumping station adjustment. Water & Wastewater Engineering., 6: 17-21.
[4] Su, Y., Wu, P.F. (2017) Application of ICEM CFD structured mesh in pump station project. Jiangsu Water Resources.,1: 13-16.
[5] Chen, H.X., Zhu, B., Li, S.B. (2010) Numerical calculation of axial flow pump model with multi-blade stagger angles. Journal of Drainage and Irrigation Machinery Engineering.,5: 378-383.
[6] Wen, D.X. (2010) Engineering Fluid Mechanics. Higher Education Press, Beijing.