Hydrodynamics Design of Multifunctional Water Tunnel

Rui Zhu¹, Shang Li¹, Yazhong Li¹, Yuan Lu¹, Feng Bao¹, Zhirong Liu*,¹

¹School of Aerospace Engineering, Xiamen University, Aerospace Building, Xiamen University Xiangan Campus, Xiangan South Road 4221-134, Xiangan District, Xiamen, Fujian, China
Email: 19122919@qq.com

Abstract. This paper studies the hydrodynamics design and analysis of a multifunctional water tunnel. On the basis of a large number of numerical simulations using the Fluent, the two-dimensional deflectors are selected and placed in the corners to streamline the water flow. The steady and streamlined section uses 10mm×90mm PVC tube as honeycomb with damping nets to obtain the low turbulence flow. The fifth power curve is selected as the contour of the contraction section after considering the flow field quality, turbulence suppression performance and velocity uniformity.

1. Introduction
The structure of the water tunnel is similar to the subsonic wind tunnel. The typical water tunnel consists of stable section, honeycomb, damping network, contraction section, experiment section, diffusion section and power system. It is suitable to study boundary layer, turbulence and fluid-structure interactions. Because the water flow is relatively uniform, and the flow visualization and quantitative PIV measurements are easier to be carried out in the water tunnel. The establishment of water tunnel has important and extensive value for the study of basic scientific problems such as turbulent flow and boundary layer[1], as well as solving practical engineering problems of civil and military. The research of this paper involves the concept design and structure design of the core components in water tunnel, the hydrodynamic design and analysis of the key sections of the water tunnel.

2. Overall Design of Water Tunnel
Overall design of the multi-function circular water tunnel is shown in Figure 1. The water tunnel mainly consists of the axial flow pump power units, the reflux pipe, the dump energy-dissipating section, the pre-contraction corner section, the steady and streamlined section, the contraction section, the experimental section, the trolley and the exit subsidence section.

The water tunnel is the horizontal circular type with open-closed experimental section. A cover plate placed on the experimental section to conduct closed operation mode with free water surface, and the open operation mode can be carried out without the cover plate. The total length of the multi-function circulating water tunnel is 18.71m, the width is 6m and the height is 3.72m (Figure 2). The experimental section size is 1m×1m×6m, the maximum flow velocity is 1m/s with the stepless speed adjustment between 0~1m/s.
In the open operation mode, there are three experimental operation combinations: the circular flow mode, the trolley drag mode and the circulation plus drag mode. The open operation mode is suitable for the experimental researches on turbulence structure and flow mechanism. The closed operation mode can improve the flow quality of the water tunnel which is more suitable for novel aerodynamics researches.

3. Hydrodynamics Design of Pre-contraction Corner Section

The installation of the pre-contraction corner section is shown in Figure 1. The deflectors with 3mm thickness made of stainless steel are used in corner section. The optimization design and selection of the deflector arrangement is carried out by numerical simulations. Table 1 indicates the case parameters for numerical simulations (D-Arc deflector, T-Straight tail, P-Horizontal plane).

| Case | D | T | P |
|------|---|---|---|
| 1    | × | × | × |
| 2    | ✓ | × | × |
| 3    | ✓ | ✓ | × |
| 4    | ✓ | × | ✓ |
| 5    | ✓ | ✓ | ✓ |

Numerical simulations of corner flow field for different selections of deflector design are carried out by the Fluent software. The numerical simulation on the corner flow field generally adopts the RNG k-ε turbulence model, the fluid medium is water and the density $\rho=998.2\text{Kg/m}^3$, the dynamic
viscosity is $\mu = 0.001003 \text{Pa s}$. The inlet boundary condition is flow velocity set as 1m/s and the turbulence is 1%. The corner flow velocity distribution nephograms of numerical simulations for 5 different selections of deflector design are displayed in Figure 3.

As the numerical simulation results shown that velocity distribution of the flow field is very uniform without obvious vortices and turbulence in Case 5. So the arc deflectors with straight tails and horizontal planes can effectively streamline the water flow, eliminate the large scale vortices and improve the quality of the flow field. The deflector deployment of Case 5 is accepted for the design of the pre-contraction corner section (Figure 4). The shape of the deflector is a quarter of the 200mm radius circle with 250mm straight tail. The front and rear edges of the deflector are made of 45° chamfering angles and the gaps of the deflectors and the horizontal planes are both 100mm.

![Figure 3. Corner flow velocity distributions](image)
4. **Low Turbulence Design of Steady and Streamlined Section**

4.1. *Design of Honeycomb*

Hydrodynamics design of the honeycomb is based on the basic principle and experience of designing the honeycomb for low-turbulence and low-speed wind tunnel [2]. The honeycomb adopts $\Phi 10\text{mm} \times 90\text{mm}$ PVC tubes and more than 90,000 PVC tubes are arranged in the steady and streamlined section. The mesh size and length of the honeycomb can meet the basic principles of hydrodynamic design and can eliminate the transverse turbulence (Figure 5).

4.2. *Optimal Design of Damping Net*

The damping net is to further improve the flow field quality by turning the large scale vortices from the honeycomb into small scale vortices, then the flow go through the subsequent contraction section whilst the flow quality is being optimized again before entering the experimental section. The damping net of the multi-function water tunnel is made of stainless steel mesh wire. The aperture ratio $R = l/d$ is the mesh diameter $l$ divided by the steel wire diameter $d$ (fixed value), and how much the turbulivity can be suppressed by different damping nets ($l/d=3, 4, 5$) with the same inlet turbulivity 0.5% under different inlet flow velocities (0.01-0.1m/s) are simulated and shown in Figure 6.

According to the numerical simulation results, as the aperture ratio increases the turbulivity suppression performance is reduced. Considering the turbulence suppression and energy loss caused by the damping nets (higher aperture ratio leads to greater flow resistance), the two 20-mesh and two 30-mesh damping nets with $R=3$ are installed in the water tunnel as shown in Figure 5.
5. Optimal Design of Contraction Section

The contraction section is between the steady and streamlined section and the experimental section, which is a transition duct with smooth curve contour. Along the flow direction, the section area of the contraction section is gradually reduced and the fluid velocity and flow field quality are continuously improved. The performance the contraction section mainly depends on the contraction ratio and contraction curve [3]. The ratio of the inlet section area to the outlet section area is the contraction ratio, and the common used contraction curves include the Witzinsky curve, the Batchelor-Shaw curve, the Bicubic curve, the optimized Bicubic curve and the fifth power curve. Figure 7 shows the numerical simulations of velocity distributions of different contraction curves with the typical inlet flow velocity 0.5m/s.
Figure 7. Numerical simulations of velocity distributions

The velocity distributions of the Batchelor-Shawv and Witozinsky curves are similar; the inlet velocities change sharply while the outlet velocities change rather smoothly. The velocity distributions of the Bicubic and optimized Bicubic curves are similar; the inlet and outlet velocities change both smoothly [4]. The fifth power curve also has the smooth inlet velocity distribution but its outlet velocity distribution is a little bit less smooth than that of the Bicubic and optimized Bicubic curves. The turbulivity suppression variations along the center axis of various contraction curves are derived
from the numerical simulation results as shown in Figure 8, for the fifth power curve the turbulivity suppression is most effective and the pressure loss is also smallest.

Based on the numerical simulation results of different contraction curves and the existing experience of water tunnel design, the fifth power curve is selected as the contraction curve where the contraction point is relatively forward at $X_m=0.5$. The length of the contraction section is 4m, the size of the contraction section inlet is $3.05m \times 3.05m$, the size of the contraction section outlet is $1m \times 1m$ and the contraction ratio is 9.3025 as shown in Figure 9. The equation of the contraction curve is:

$$\frac{R-R_2}{R_1-R_2} = 1 - 10 \left(\frac{x}{L}\right)^3 + 15 \left(\frac{x}{L}\right)^4 - 6 \left(\frac{x}{L}\right)^5, X_m = 0.5$$

(1)

6. Performance Evaluation of Experimental Section

The experimental section of the water tunnel is the key operation section for the flow simulation and PIV quantitative measurements. The design of the experimental section requires that the velocity and direction of water flow are evenly distributed in the space and capable of providing good flow quality with low initial turbulence [5]. The experimental section structure adopts the right angle design which can effectively eliminate echo and ensure the quality of flow field. The length of the experimental section is 6m, the size of the entrance and exit cross-section is $1m \times 1m$ as shown in Figure 10.

6.1. Performance Evaluation of Open Mode

The water level in experimental open mode is set at 0.95m, and the velocity distributions of the middle cross-section of the experimental section ($x=7.5m$) are calculated with various outlet velocities of the contraction section (0.1m/s, 0.2m/s, 0.5m/s, 0.8m/s). Figure 11 shows the mesh model for the numerical simulations of open mode, the total length of the contraction section is 4m and the height of the water level is equal to that of the experimental section. The size of post-contraction section is $1m \times 0.95m \times 6.5m$ (with 0.5m transition section), the upper surface is sliding wall surface and the other three surfaces are fixed wall surfaces in numerical calculations.
Figure 12. Velocity distributions of middle cross-section ($x=7.5\text{m}$, $y=0.5\text{m}$) (open mode)

Figure 13. Spanwise velocity distributions (open mode)

The trends of the velocity distributions of the middle cross-section under different flow velocities ($0.1\text{m/s}$, $0.2\text{m/s}$, $0.5\text{m/s}$, $0.8\text{m/s}$) are similar as shown in Figure 12. In open test mode, the uniform
flow velocity distributions can be established within a rather short distance behind the contraction section outlet in the range of designed flow velocities (0-1m/s) after going through the three-surface contraction. As Figure 13 shows the spanwise velocity distributions of the middle cross-section of the experimental section are relatively uniform, and the flow field quality of the experimental section is good in open mode.

6.2. Performance Evaluation of Closed Mode

The water flow is compressed by all four inner surfaces of the contraction section in closed mode, and the mesh model for the closed mode numerical calculations is shown in Figure 14. The size of post-contraction section is 1m×1m×6.5m (with 0.5m transition section) and all four inner surfaces are fixed wall surfaces in numerical calculations. The trends of the velocity distributions of the middle cross-section under different flow velocities are also similar in closed mode (Figure 15), the uniform flow velocity distributions can also be established within a rather short distance behind the contraction section outlet and with more smooth velocity distributions inside the contraction section. Figure 16 shows the spanwise velocity distributions of the middle cross-section which exhibit the stable and uniform flow quality that is well accepted for the closed mode experiments.

![Figure 14. Mesh model of closed mode](image-url)
Figure 15. Velocity distributions of middle cross-section ($x=7.5m$, $y=0.5m$) (closed mode)

Figure 16. Spanwise velocity distributions (closed mode)

7. Conclusions
The arc deflectors with straight tails and horizontal planes are selected and the shape of the deflector is a quarter of the 200mm radius circle with 250mm straight tail in the pre-contraction corner section. The front and rear edges of the deflector are made of 45° chamfering angles and the gaps of the deflectors and the horizontal planes are both 100mm.

The honeycomb adopts $\Phi 10mm \times 90mm$ PVC tubes and more than 90,000 PVC tubes are arranged in the steady and streamlined section. According to the numerical simulation results, as the aperture ratio increases the turbulivity suppression performance is reduced. Considering the turbulence
suppression and energy loss caused by the damping nets, the two 20-mesh and two 30-mesh damping nets with R=3 are installed in the steady and streamlined section.

Based on the numerical simulation results of different contraction curves, the fifth power curve is selected as the contraction curve where the contraction point is relatively forward at $X_m=0.5$. The length of the contraction section is 4m, the size of the contraction section inlet is $3.05m \times 3.05m$, the size of the contraction section outlet is $1m \times 1m$ and the contraction ratio is 9.3025.

The uniform flow velocity distributions can be established within a rather short distance behind the contraction section outlet in the range of designed flow velocities (0-1m/s) for both open and closed experimental modes. The results of performance evaluations indicate that the flow field quality of the experimental section is good in both open and closed mode.

8. References
[1] Daniel, Libin 2015 Design of a recirculating water tunnel for the study of high Reynolds number turbulent boundary layers ASME International Mechanical Engineering Congress and Exposition
[2] Ding, S M 2015 Design and experiment of NJS-1 type open-circuit closed wind tunnel for plant protection Transactions of the Chinese Society of Agricultural Engineering 31.4 76-84 (9)
[3] Madariaga and Gabriel 2006 Anti-cavitation system in pipelines which avoids that the fluid reaches its vapour pressure at the output of a given contraction using a device that connects the output section of the contraction with its downstream pressure US, US20060243336
[4] Arai, Kunio, Takahashi K and Saito S 2006 Correlation of velocity distributions in an anchor-agitated vessel using a bicubic b-spline function Journal of Chemical Engineering of Japan 15.5 383-9
[5] Tritico, Hans M and Hotchkiss R. H 2005 Unobstructed and Obstructed Turbulent Flow in Gravel Bed Rivers Journal of Hydraulic Engineering 131.8 635-45