Numerical Simulation of Hydraulic Characteristics of Vertical Lift Gate for Different Openings

Shihua He[a], Tingting Yang[b] and Chunying Shen[c]

Faculty of Electric Power Engineering, Kunming University of Science and Technology, Kunming, 650500, China

[a]hoxiwa@aliyun.com, [b]912832062@qq.com, [c]86295700@qq.com

Key words: vertical lift gate, numerical analysis, vortex structure, gate opening, hydraulic characteristics

Abstract: Based on establishing the computational fluid domain of underflow vertical lift gate under different openings, the 3D numerical simulation is carried out by a combination of VOF model and standard κ-ε model to predict the distribution of flow velocity, pressure and turbulence kinetic energy. The results show that there are different sizes of vortexes in the downstream of the gate due to flow separation from the gate bottom edge. The smaller the gate opening is, the greater the flow velocity underneath the gate is, and the further away from the gate the vortex structure is. Turbulent kinetic energy decreases with the increasing of gate openings and impacts the flow state in the vicinity of gate. The smaller the opening is, the greater the pressure difference between the upstream and downstream of the gate is. The negative pressure mainly appears on the downstream of the gate. Appropriate increasing the gate opening is conducive to the safe and stable operation of the gate.

Introduction

The vertical lift gate is a kind of important hydraulic structures mounted in a flow passage to adjust the discharge magnitude and water level by its opening and closing, which is widely used in agricultural irrigation, flood control, and hydroelectric power, etc. The hydraulic characteristics is refers to the distribution and change of physical variables such as the velocity, pressure, turbulent kinetic energy and its dissipation rate of water flow in the vicinity of a lift gate. With the adjustment of the gate openings, the hydraulic characteristics of gate also produce very big change, which may result in the occurrence of gate vibration phenomenon.

In addition to the hydraulic model test, numerical simulation has become an effective method to predict the flow field around the gate. Kostecki [1-3] computed the two-dimensional flow in the vicinity of gate using combinative modelling of the vortex method and the boundary element method. Kazemzadeh-Pardi [4] applied smoothed fixed grid finite element method to solve free surface flow in gated tunnels. Liu [5] simulated the turbulent flow behind a sluice gate with submerged discharge conditions. Pani et al. [6] used finite element technique to investigate the effect of fluid-structure interaction on the hydrodynamic pressure. In the present work, a numerical method, being a combination of VOF model and standard κ-ε model, is applied to predict the distribution of flow velocity, pressure and turbulence kinetic energy in the vicinity of a vertical lift gate for different openings.

Numerical Models

VOF Model and Governing Equations. Due to non-pressure flow condition on the downstream of the gate, VOF model is used to track the free surface. There are both the air and the water phases in the model. Assuming that α_q denotes the volume fraction function for phase q. $\alpha_q=0$ indicates excluding phase q in the region, $\alpha_q=1$ indicates that the region is full of phase q, and $0<\alpha_q<1$ indicates including both the phase q and other phases. The governing equations of α_q are as follows:

$$\frac{\partial \alpha_q}{\partial t} + u_i \frac{\partial a_q}{\partial x_i} = 0 \qquad (1)$$

where t is time and u_i is the mixing fluid velocity component in x_i-direction.

The standard κ-ε turbulent model is applied to simulate the flow field. The instantaneous continuous equation, Navier-Stokes equations and standard κ-ε equations which are embeded VOF model are as follows:

Continuous equation:

$$\frac{\partial r}{\partial t} + u_i \frac{\partial (r u_i)}{\partial x_i} = 0 \qquad (2)$$

Navier-Stokes equations:

$$\frac{\partial (r u_i)}{\partial t} + \frac{\partial (r u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}[(m + m_t)(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i})] \qquad (3)$$

Turbulent kinetic energy κ-equation:

$$\frac{\partial (r k)}{\partial t} + \frac{\partial (r k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}[(m + \frac{m_t}{s_k})\frac{\partial k}{\partial x_j}] + G_k - r e \qquad (4)$$

Dissipation rate ε-equation:

$$\frac{\partial (r e)}{\partial t} + \frac{\partial (r e u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}[(m + \frac{m_t}{s_e})\frac{\partial e}{\partial x_j}] + \frac{C_{1e} e}{k} G_k - C_{2e} r \frac{e^2}{k} \qquad (5)$$

where ρ is the specific mass of mixing fluid, p is pressure, and $G_k = m_t(\partial u_i/\partial u_j + \partial u_j/\partial u_i)\partial u_i/\partial u_j$, $m_t = r C_m k^2 / e$, $C_m = 0.09$, $C_{1e} = 1.14$, $C_{2e} = 1.92$, $s_e = 1.3$, $s_k = 1.0$.

Meshing and Boundary Conditions. The whole computational model consists of pressured section, gate section and non-pressure section which are all in a rectangle tunnel. The height and width of tunnel are 2 m, and the simulated lengths before and after the gate are 7m and 15m respectively. The thickness of the gate is 0.2m, and the openings are set respectively for 0.1m, 0.2m, 0.4m and 0.6m. The hexahedral element mesh is adopted to generate the computational domain. Fig.1shows a meshing schematic which has 589240 mesh elements

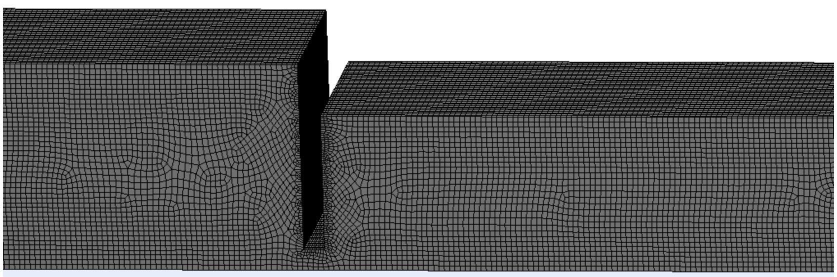


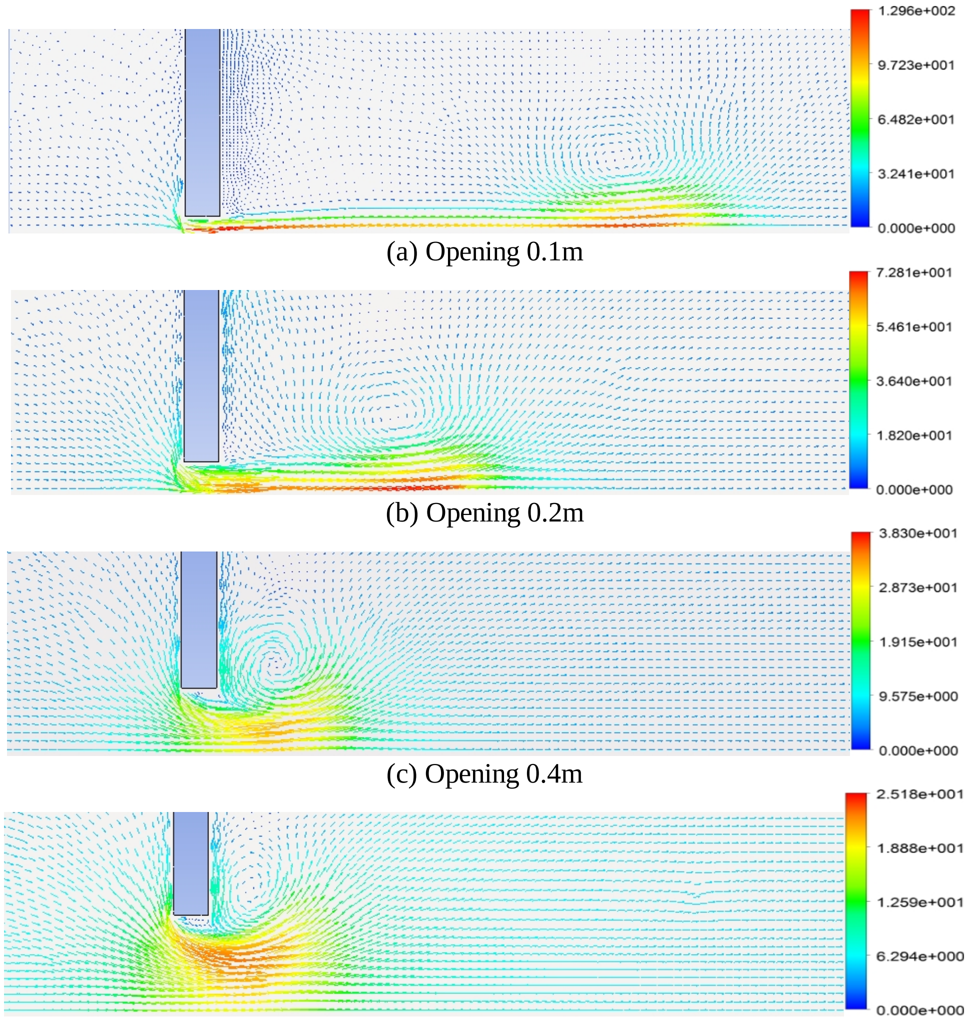
Fig.1 Meshing schematic of computational domain

The boundary conditions of whole computational domain include inlet, outlet, water-air interface on the downstream of gate, and solid wall boundaries. The flow inlet is set for velocity boundary. The average velocity at inlet section is calculated by $u_0 = Q/BH$ where Q=20m^3/s, B is the inlet width, and H is the inlet height. The flow outlet is set for pressure boundary and the value is one standard atmospheric pressure. The VOF model is used to deal with the water-air interface on the downstream of gate. The solid wall boundaries are no slip conditions and the given normal velocity is zero. The wall function method is applied to process the near wall viscous sublayer.

Results and Discussion

Selecting the flow direction is the *x*-axis, the vertical direction is the *y*-axis, and the horizontal direction is the *z*-axis. The *xoz*-plane is fixed at the bottom of the tunnel, the *yoz*-plane and the *xoy*-plane is fixed at the middle sections of the gate and tunnel respectively. The step length of time is $\Delta t = 0.001\text{s}$.

Fig. 2 shows the flow velocity vector diagrams in the vicinity of the gate orifice at $t = 0.06\text{s}$ and *xoy*-plane for different openings. It can be seen from Fig. 2 that the vortex is generated in the downstream of the gate. The flow before the gate is relatively stable. When water flows through the gate orifice, it detaches from the bottom edge of gate and the velocity is rapidly increasing. There are the recirculation zone and vortex behind the gate. It can be observed in Fig. 2 that the smaller the gate opening is, the greater the flow velocity through the gate orifice, and the further away from the gate the vortex structure is. At the same time, the diameter of the vortex structure grows with the decreasing of the gate opening due to vortex diffusion and mixture with the downstream flow.



(a) Opening 0.1m

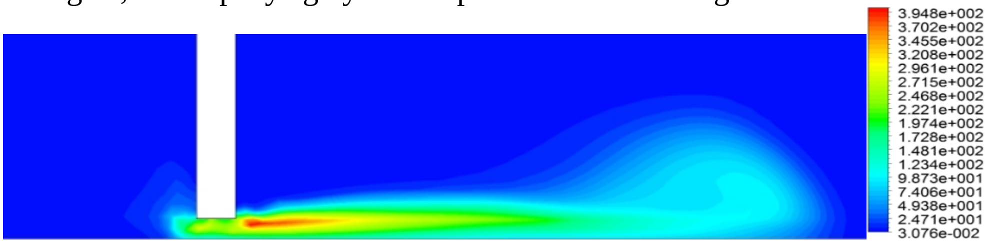
(b) Opening 0.2m

(c) Opening 0.4m

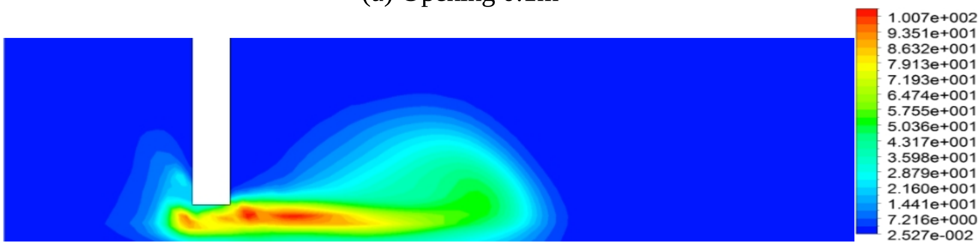
(d) Opening 0.6m

Fig. 2 Velocity (m/s) vector distributions for different gate openings

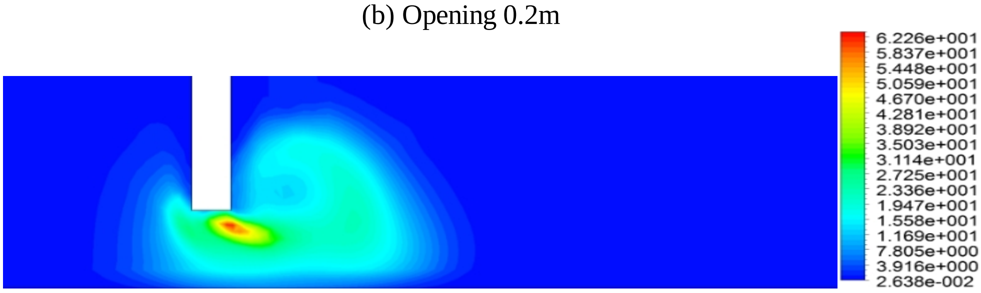
Fig. 3 shows the turbulence kinetic energy distribution nephograms at $t = 0.06\text{s}$. It can be seen in Fig. 3 that the turbulent kinetic energy is greater around the gate orifice, and is smaller on both sides of the gate. The maximum turbulent kinetic energy decreases as the gate opening increases. Meanwhile, with the increasing of the gate openings, the maximum turbulent kinetic energy moves gradually from the rear trailing edge of the gate towards the bottom of the gate. The distribution differences of turbulence kinetic energy result in the complication of the flow regime in the bottom and the downstream of the gate, accompanying by flow separation and vortex generation.



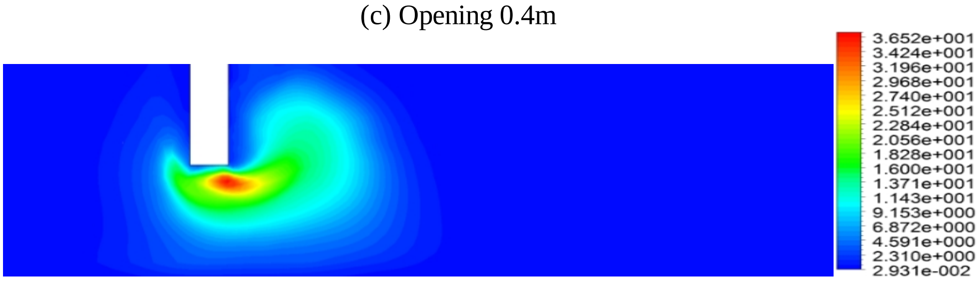
(a) Opening 0.1m



(b) Opening 0.2m



(c) Opening 0.4m



(d) Opening 0.6m

Fig. 3 Turbulence kinetic energy distributions for different gate openings

Fig. 4 shows the pressure change curves at $y = 0.05\text{m}$ in *xoy*-plane and $t = 0.06\text{s}$. It can be seen from Fig. 4 that the maximum value of the pressure all appears on the upstream side near the gate, and the minimum value appears in the gate back for four different openings. The smaller the gate opening is, the greater the pressure difference between the upstream and downstream of the gate is. The pressure distribution is more uniform with the increasing of gate opening. The maximum negative pressure occurs on the downstream when the gate is minimum opening.

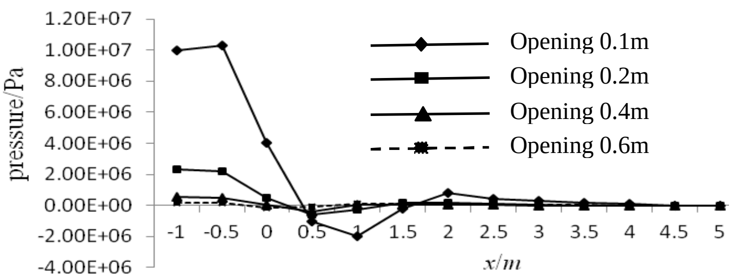


Fig. 4 Pressure change curves for different gate openings

Conclusions

In the present work, the ANSYS Workbench is used to establish the computational fluid domain and the Fluent software is used to calculate the flow parameters. The distributions of velocity, turbulence kinetic energy and pressure are obtained respectively. The main conclusions are as follows. the smaller the gate opening is, the faster the vortex structure on the downstream of the gate moves towards the outlet, and the larger the vortex diameter is, which affects the distribution of velocity and pressure field. Meanwhile, smaller gate opening results in greater pressure difference between the upstream and downstream of the gate and more prone to generating negative pressure, which may cause the cavitation and vibration of the gate. The increasing of the gate opening can obtain a more uniform pressure distribution and benefit the safe and stable operation of the gate.

Acknowledgements

Financial support provided by the National Natural Science Foundation (51369013) is gratefully acknowledged.

References

[1] S. W. Kotecki, Numerical modelling of flow through moving water-control gates by vortex method. Part I-problem formulation [J]. Archives of Civil and Mechanical Engineering, VIII (3) (2008): 73-89.

[2] S. W. Kotecki, Numerical modelling of flow through moving water-control gates by vortex method. Part II-calculation result [J]. Archives of Civil and Mechanical Engineering, VIII (4) (2008): 39-49.

[3] S. W. Kotecki, Numerical analysis of hydrodynamic forces due to flow instability at lift gate [J]. Archives of Civil and Mechanical Engineering, XI (4) (2011): 943-961.

[4] M. J. Kazemzadeh-Parsi, Numerical flow simulation in gated hydraulic structures using smoothed fixed grid finite element method [J]. Applied Mathematics and Computation, 246(2014): 447–459.

[5] S. H. Liu, T. T. Liao, Q. S. Luo, Numerical simulation of turbulent flow behind sluice gate under submerged discharge conditions [J]. Journal of Hydrodynamics, 27(2) (2015): 257-263.

[6] P. K. Pani, S. K. Bhattacharyya, Hydrodynamic pressure on a vertical gate considering fluid-structure interaction [J]. Finite Elements in Analysis and Design, 44(2008): 759-766.