Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and 8th International Conference on Nanochannels, Microchannels, and Minichannels
FEDSM-ICNMM2010
August 1-5, 2010, Montreal, Canada

FEDSM-ICNMM2010-30058

STUDY ON IMPROVING THE FLOW IN FOREBAY OF THE PUMPING STATION

Chao Liu, Jiren Zhou, Li Cheng, Yan Jin
College of Hydraulic Engineering, Yangzhou University
131 Jiangyang Middle Road, Yangzhou 225009, China
E-mail: liuchao@yzu.edu.cn; zhoujr@yzu.edu.cn; chengli@ yzu.edu.cn; jinyan_yu@163.com
Xu Han
Institute of Design and Research on Hydraulic Engineering of Inner Mongolia
85 Hulen South RD., Huhehaote, Inner Mongolia
E-mail:hanxu_hx@163.com

ABSTACT
The flow pattern in the forebay of pumping station is of very important influence on the operation of the pumps. The different kinds of unlike flow pattern, such as large scale circulations and vortices, are often found in the forebay of pumping station especially with multi-pumps. This paper takes the study on improving the flow pattern in the forebay to ensure a good flow condition for pumps operation. The computational results of three dimensional flow fields in an original designed forebay were obtained and a large circulation was found in the forebay on the side which caused serious problems to the operation of pumps. A typical combined means were applied to improve the flow motion. The computational results of the flow fields shows that the large circulation was eliminated and the flow distribution was improved, which was verified by the model test. The improved design has been successfully used to a large pumping station.

Key words: Forebay, Pumping station, Flow pattern, Numeric simulation, vortex

NOMENCLATURE

C_μ, C_1, C_2 – empirical constants,
$C_{1\varepsilon}$, $C_{2\varepsilon}$ – empirical constants,
E – roughness,
g_i – volume force components,
h_p – the height of sill,
h – water depth,
k – turbulent kinetic energy,
l_v – the length of vortices,
L – the feature length of turbulent flow,
p^* – equivalent pressure,
R_e – Reynolds Number,
u_i, u_j – velocity components calculated, represent u, v, w,
u_τ – Shear Velocity,
u_{in}, – flow velocity at inlet of the domain along x axis,
x_i, x_j – axes in rectangular coordinates, represent x, y, z;
y_p – distance from the node to wall surface,
y^+ – feature distance $y^+ = y_p U_\tau/\nu$,
ε – dissipation rate,
κ – Carman constant,
ρ – fluid density;
μ_e – effective viscosity coefficient,
μ_t – turbulent eddy viscosity coefficient.

Usually the width of the pumping intake forebay with multi-units is larger than the width of the intake river. The transition passage between pumping station and Intake River forms a diffusion passage so-called diffusion forebay. In case of the width of some pumping intake forebay is equal to the width of the intake river the diffusion passage is not necessary. The intake channel near the pumping station can be considered a part of forebay. The intake forebay with lateral water flow has the characteristics of flow in curved channel. The water keeps the flow pattern of first contraction and then diffusion. In the process of diffusion flow the vortex and backflow are formed. These backflow and vortex effect the flow fields in the pump sump or suction passage which may causes the strong surface vortices and submersible vortex tubes which will result in the serious vibration of pump units and damage to the operation of pumping system. Therefore the only way to ensure the normal operation of the pump unit is eliminating the vortex, backflow and keeping a good velocity distribution at the pump inlet.

In the most studies of such topic the model test is conducted through the establishment model test. Since the limitation of measurement conditions of model test, one cannot get all the flow field of the physical model. To further develop the research of this aspect, the numerical simulation is made through commercial software platform to intestate the 3d flow movement in a lateral intake forebay in this paper. The 3d flow fields are obtained and are consistent with the experimental results.

1. MATHEMATICAL MODEL

Intake river water pumping station is within a fully developed turbulence, as a constant flow movement, only when you examine the movement of both the situation does not take into account the changes in flow over time, its flow control equation of motion is:

1.1 Continuity equation

$$\frac{\partial(\rho u_j)}{\partial x_j} = 0 \qquad (1)$$

1.2 Motion equation

$$\frac{\partial(\rho u_j u_i)}{\partial x_j} = \rho g_i - \frac{\partial p^*}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu_e\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] \qquad (2)$$

Where ρ is fluid density; x_j is the coordinates x，y，z; u_i is the components of average relative velocity; u，v，w; ρg_i is the components of volume force; p^* is equivalent pressure; μ_e is effective viscosity efficient, $\mu_e = \mu + \mu_t$, μ is molecular viscosity efficient, μt is turbulent eddy viscosity efficient.

1.3 Turbulence Model

In order to determine the effective viscosity μe the turbulent eddy viscosity μt must first determined. RNG *k-ε* turbulence model based on the transport equation with turbulence k and its dissipation rate ε is applied here. In RNG *k-ε* turbulent viscosity model turbulent viscosity is revised, and the average flow circulation is considered which is adequate to processing the flows of a large degree of bending, and so more suitable for numerical simulation of the complex flow like side intake flow.

k-ε turbulent model including *k* equation and *ε* equation：

$$\frac{\partial}{\partial x_j}\left[\rho u_j k - \left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] = \rho(P_k - \varepsilon) \qquad (3)$$

$$\frac{\partial}{\partial x_j}\left[\rho u_j \varepsilon - \left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] = \rho\frac{\varepsilon}{k}(C_1 P_k - C_2\varepsilon) \qquad (4)$$

Where P_k is produced from turbulent kinetic energy, presented as:

$$P_k = \frac{\mu_t}{\rho}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} \qquad (5)$$

μt is calculated from next equation：

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \qquad (6)$$

Selecting a number of empirical coefficients in turbulence model directly affects on the calculation results. For the complex flow with a free surface, the coefficient value is particularly important, which requires a large number of calculations in accordance with test verification to be rationally determined.

2. BOUNDARY CONDITIONS

2.1 The inlet boundary

The inlet of calculated flow field located in the upstream far enough to outlet of intake channel. The flow at cross-section of inlet is already fully developed turbulence flow and more uniform. The flow velocity components at *y* and *z* direction in this cross-section are zero. Velocity at *x* direction $u=u_{in}$ (u_{in} known as flow rate of inlet). Unit turbulence kinetic energy *k* can take as $0.005u_{in}^2$, take the turbulent dissipation rate ε as $C_u k_{1.5}$ / L, C_u = 0.09, L is the turbulence special length, represented by water depth h.

2.2 The outlet boundary

Outlet flow is similar to the inlet flow and has become a one-way. Without consideration for the effects on calculation regional solution of the downstream of outlet, the outlet boundary conditions is that

$\partial u/\partial y = 0$，$\partial P/\partial y = 0$.

and k、ε taking value same as the inlet.

2.3 Free surface

The flow velocity within the intake channel is slow and its surface fluctuation is small. The calculation according to symmetric boundary conditions, namely, all the variable gradients along z direction are zero:

$$\frac{\partial \phi}{\partial z} = 0 \qquad (7)$$

where ϕ is ui，k，ε and so on.

3. SIDE WALL

There is viscous sub layer near-wall region in which turbulence development is not adequate, low Reynolds number, large velocity gradient; the k-ε turbulence model can not be used in this region. The Law of the wall is applied hear to calculate the velocity near the wall, as the iterative calculation of the boundary conditions:

$$u^{+} = \frac{1}{\kappa}\ln(Ey^{+}) \qquad (8)$$

where $u^{+}=u/u_{\tau}$, u_{τ} is the rate for the resistance; $y^{+}=y_{p}u_{\tau}/\nu$, y_{p} is the node-to-wall distance; κ is Carmen constant; E is roughness constant, for the smooth wall take E = 9;

Taking $k = u^{2}/C_{u}^{0.5}$ and $\varepsilon = u^{3} / (kY)$. As the velocity gradient in the viscous sub layer is large, so the calculation grids are increased near the wall.

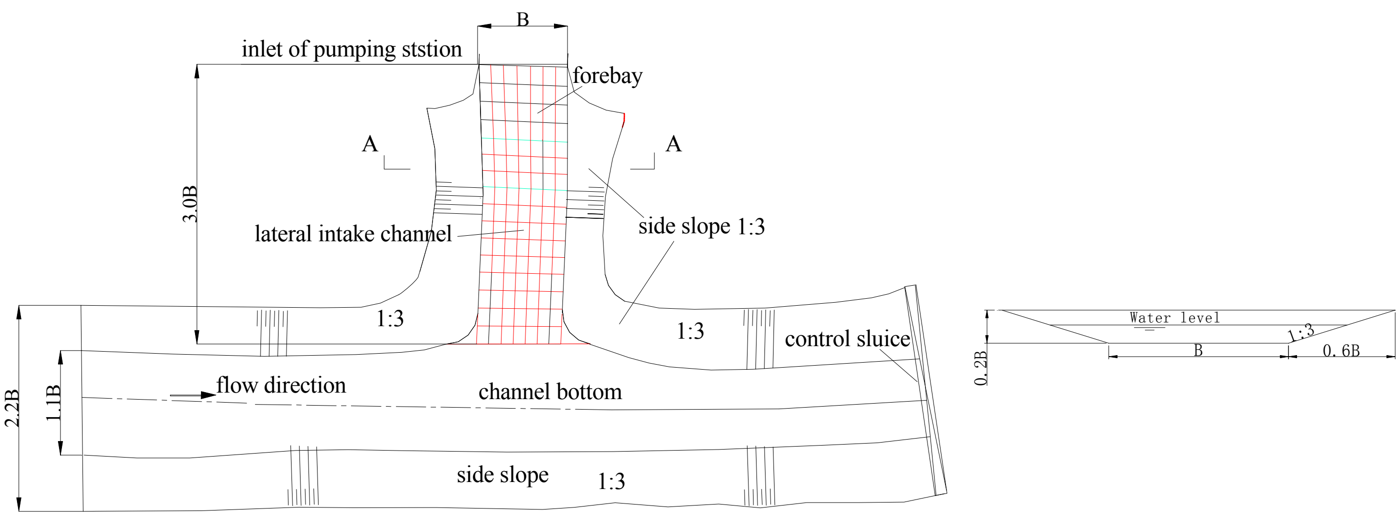
The Reynolds Number R_e is within the zone of resistance square.

4. NUMERICAL SIMULATION

Bi-lateral intake channel is used for the pumping station, The design pumping flow rate is 180 m^3 / s, the design lift is 6.0m, the higher intake water level is 0.7m, the lower intake water level is 0.3m. Model scale is 1:50, the calculation region of the model is shown in Figure 1.

Since the model structure is more complex and irregular, sub-block partition and local densification technology are applied here for solid mesh grid generation. The structured grids with hexahedron shape were used in regular regions, and the unstructured grids with tetrahedron shape were used in complex regions. Taking into account the capacity of the computer and the time-consuming for calculation, the grids sizes of calculation for different alternatives are 400,000 ~ 600,000 units and a steady state calculation for the model was conducted.

We know that pump has a greater impact on the upstream of forebay when the pump units are partly operated especially with asymmetric operation resulting in recirculation in the forebay. These cases are considered in the test alternatives with comprehensive and effective means to improve the flow pattern in the forebay.



(a) (b)

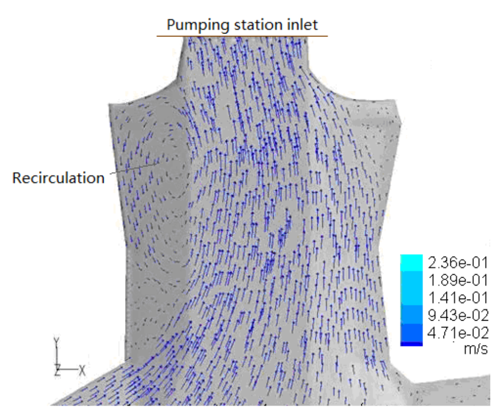
Fig.1 Plan and section view of pumping station intake channel

(a) Plan view of the intake channel (b) Section view of the intake channel

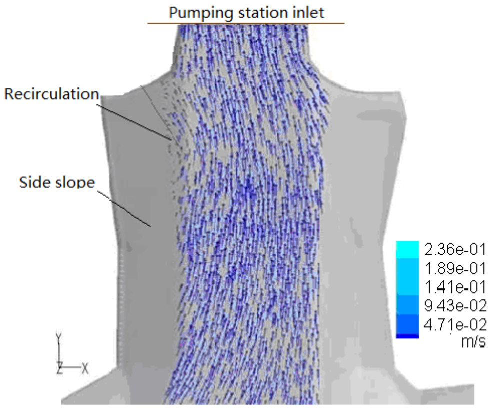
4.1 Numerical Simulation of the original design

In the original design the width of the pumping station intake channel keeps constant with a side slope of 1:3. The calculation region includes model intake channel and suctions passage. The designed water levels are considered for the simulation. According to the model characteristics and the calculation conditions, considering the structure of the whole model more complex, the mixed structures of the grids are applied for partitioning the computational region with block subdivision. The minimum edge angle of the grid is 21degree and the quality of the mesh grids are satisfactory to the calculation.

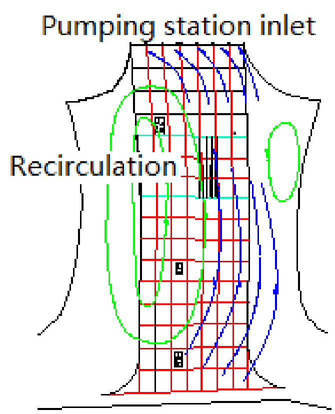
The normal water level in the calculation of flow field conditions as shown in Figure 2. In Figure 2 (a) is a surface layer distribution of flow velocity vector at a plane section of intake channel (5cm underwater surface). From the figure we can see that there is large-scale surface recirculation at the left of pumping station. The water returns before wing-wall along the side slope of the channel and format recirculation zone, most of which is above the area of the intake channel slope, the range about one fifth of width of the intake channel Bottom are affected by recirculation.



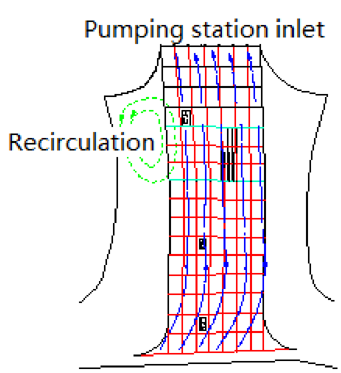
(a)



(b)



(c)



(d)

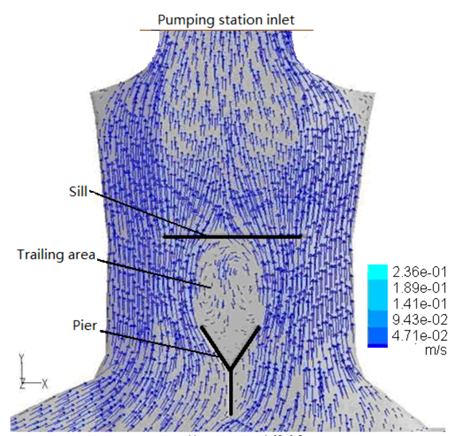
Fig.2 Velocity field and flow pattern of original scheme
(a)Calculated velocity distribution of surface layer
(b) Calculated velocity distribution of bottom layer
(c) Measured and observed flow pattern of surface layer
(d) Measured and observed flow pattern of bottom layer

From Figure 2 (b) it can be seen that recirculation also exists in the bottom layer which affects water distribution in front of pumping station. The deflection of bottom flow occurs from right to left. In the entrance are skewed to the left which can easily cause vortex at Inlet. In addition, there is a small recirculation zone in front of the right wing wall.

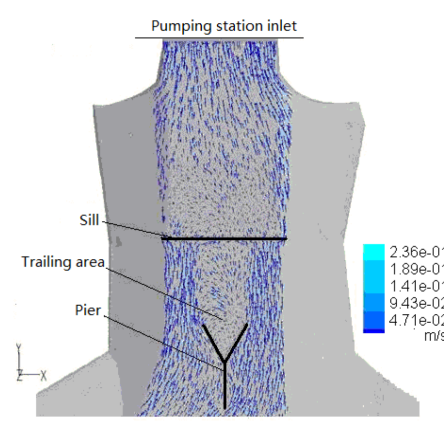
Figure 2 (c) and Figure 2 (d) are the flow pattern of intake channel obtained from a model test observations. It can be seen from the graph that flow trend are basically the same as the trend observed from model test. Because the model experimental observation is only an intuitive observation of the flow pattern, the detail change in recirculation is difficult to describe accurately. While the calculated result is a complete, quantitative flow field, numerical simulation, therefore, compensate for the lack of experimental observations. The model experiment is verification to the numerical simulation results.

4.2 Numerical Simulation of improving design

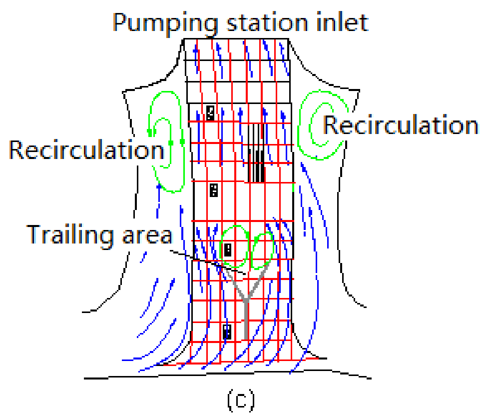
From the numerical simulation results it can be seen that in the original design, both surface flow and bottom flow of intake channel have recirculation. Oblique flow into the inlet conduit, side-flow deflection is particularly serious, apt to swirl, resulting in hydraulic vibration and power increase, less efficient units, while heavy silting which raised the riverbed in the intake channel affects on pump safety of operation. In order to ensure good water conditions, It is necessary to take engineering measures for improving the flow pattern. On the basis of a large number of studies a combined flow-adjust measures “sill + pier" is used to improve the flow pattern. The satisfactory results are obtained. Combined flow adjustment design is shown in Figure 3 and Figure 5.



(a)



(b)



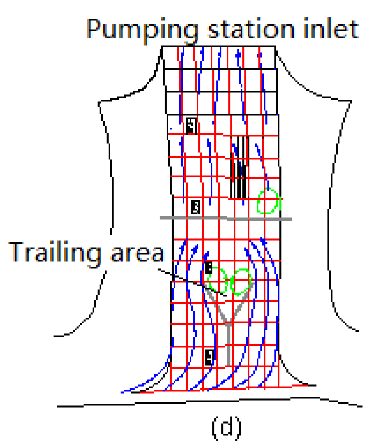


Fig.3 Velocity field and flow pattern of improved scheme

(a)Calculated velocity distribution of surface layer
(b) Calculated velocity distribution of bottom layer
(c) Measured and observed flow pattern of surface layer
(d) Measured and observed flow pattern of bottom layer

In the normal water level conditions, after the addition of Y-shaped pier the water flow is divided into two parts of the main flow, the large-scale recirculation disappeared. in spite of the vortices near the wing walls on both sides do exist, the vortex scale significantly reduced. The water at the front of the pumping station flows basically along the longitudinal direction, shown in Figure 3 (a) as shown.

In the normal water level conditions, after the addition of Y-shaped pier the bottom water is also divided into two parts of the main flow and rationalizes the distribution of the entire flow. Meanwhile greater trailing vortices occur in the region after the pier. The water (including the trailing vortices) passing over the sill results a horizontal rolling after the sill. So the water flow receives the second adjustment. Through consecutive adjustments the flow at inlet of the pumping station has been straight, shown in Figure 3 (a), Figure 3 (b). Figure 3 (c) h, Figure 3 (d) shows the flow pattern of the test observations. The test results agree with the calculated results.

Usually velocity distribution uniformity (V_u) is used to measure the velocity distribution level; V_u is determined by the following formula:

$$V_u = \left[1 - \frac{1}{\overline{u_a}} \sqrt{\frac{\sum (u_{ai} - \overline{u_a})^2}{m}} \right] \times 100\% \qquad (5)$$

Where: $\overline{u_a}$ is average velocity of the calculated cross-section; u_{ai} is the flow velocity at i th point of the flow cross-section; m is number of the points within the cross-section of the flow.

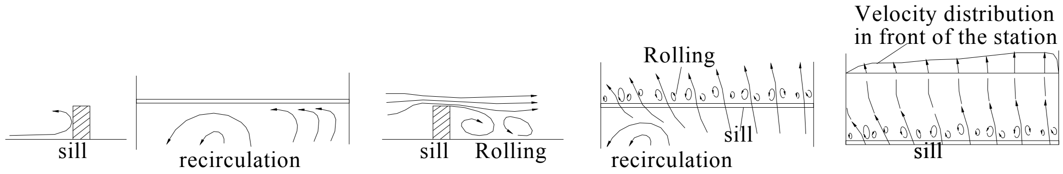
Based on flow field data obtained from the numerical simulation for the improving design the velocity distribution uniformity 10cm before the Inlet section is calculated as 84.43% which indicates good uniformity of velocity distribution. Of course, we should consider the direction of flow because the flow rate uniformity does not reflect the flow angle. The angle between flow direction and longitudinal direction is very small, flow line is straight, which can meet the requirements of pumping station operation.

The flow pattern in front of pumping station has been significantly improved after adding the sill and pier. The pier play a role of dividing the water flow in two parts, forcing water flows on to both sides of wall and provide adequate water flow momentum to overcome the backflow and separation, so that water flow evenly spreads. Meanwhile, the water flow passing through the pier and sill result in additional hydraulic losses.

5. ANALYSIS AND DISCUSSION

5.1 Sill Flow Adjustment

Among a variety of Flow Adjustment means sill is one of the measures with the simplest structural form convenient in construction, and good in adjustment effect. It has been now used to improve flow pattern in a number of pumping station [2] ~ [5]. Through vortex motion, flow mixing, the sill results in flow adjustment and energy exchange, so that the original water flow structure is completely changed, so as to achieve the purpose of improving the flow pattern. The process of flow adjustment by sill is shown in Figure 4 which includes blocking the main flow near the bottom; sill top separation; eddy mixing and flow redistribution



(1) (2) (3) (4)

Fig.4 Four stages of the flow over passing the bottom sill

(1) Cutting bottom flow, (2) Separation at sill top, (3) Turbulent mixing, (4) Flow redistribution

(1) The sill is located in the recirculation zone at appropriate location which directly blocks the main flow near bottom so that the initial flow patterns can not be sustained. When the adjusted main flow reach the sill it is prevented and the flow kinetic energy changes into pressure energy leading to pressure increase in main flow region and forcing the main flow to move along the sill horizontally into non-main flow regions, which results in the first flow redistribution.

(2) Then the water flows upward and climbs over the sill and separates at the side edge of sill, consequently forms vortex rolling behind the sill. In the vortex rolling area there are full of vortices.

(3)Through turbulent mixing in the vortex rolling region flow kinetic energy can be exchanged, at the same time accompanied by mass exchange, which results in second flow redistribution.

(4) Then water leaves the vortex rolling area to downstream. At a certain distance after the sill the turbulence of flow fully developed and so the velocity distribution tends to uniformity. Over-sill-flow is different from the single forward-step-flow or single backward-step-flow, and also different from the combinations step flow, because the top of the sill is very narrow there is no enough space over top for water to develop. In the process of over-sill-flow, the water flow structure drastically changed, and accompanied by a strong energy exchange. The hydraulic losses are mainly caused by the vortex rolling after sill. Generally one believes that the energy dissipation of vortex roll-area is large. However both the physical model tests and pumping station site tests show that the hydraulic losses are very small which is similar to the plate valve in pipeline with a particular flow loss. This phenomenon shows that the vortex rolling formed in the over-sill (valve)-flow is orderly vortex motion. The energy dissipation in the process of energy exchange is not large, so the hydraulic losses caused by sill are very small.

The cross-sectional geometry shape of the sill can be rectangular or trapezoidal with side edges, so that location of flow separation point is fixed, The area of vortex rolling is relatively stable. Here a rectangular cross section is adopted for the sill. The relative sill height (sill height and water depth ratio H_{sill} / h_w) affects the calculation of flow field. The influences of different sill height on calculated velocity distribution uniformity at the inlet are shown in Table 1. The location of cross section is 10cm far from the inlet of pumping station.

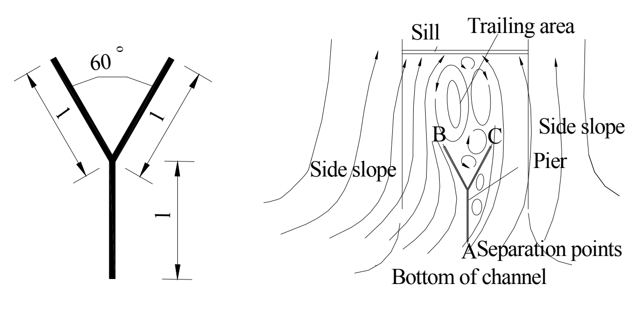
Table 1 The influences of the sill heights on the velocity distribution uniformity of calculation

Sill relative height H_{sill}/h_w (%)	working conditions	sectional velocity distribution uniformity before pumping station V_u (%)
0.3	Normal water	76.1
0.4	Normal water	78.3
0.50	Normal water	73.6

As seeing from the table 1, the relative height of sill affects on the flow pattern after sill. H_{sill} can be taken 0.30 ~ 0.45 times of normal depth(h_w), that is H_{sill}/h_w = (0.30 ~ 0.45). A better adjustment results could be obtained.

5.2 Diversion pier flow adjustment

The process of diversion pier adjustment to flow pattern is similar to the flow around prism (Figure 5a cylinder ABC). When the water flow passing the pier, reaction force of positive side surface AB makes flow turning back to the inside of bend flow and eliminate a large-scale recirculation. At the same time, when flow diversion around piers flow separation occurs at A, B, C, and the flow separation point is fixed. This is the characteristics of the flow around prism movement. The fixed separation point can guarantee the stability of around-flow pattern therefore to ensure the stability of the entire movement.



(a) (b)

Fig5 Flow pattern around the diversion pier

(a) Plan of diversion pier, (b) Observation of flow pattern around the pier

After the separation from point A the flow generated trailing vortices behind the AC surface. Due to the oppression of the main flow the trailing vortices is small. When water passes through and produces separation at the points B, C a large trailing vortices is observed behind the BC side. This is because along the main flow direction there is sufficient spaces for trail developing. Compared with the original design, after adding the piers, the main flow is split into two shares. the large-scale recirculation zone at one side of intake channel becomes ones located in the middle with small size. So that flow pattern has important change: flow symmetry is significantly improved and intensity of vortex is reduced. This provides an effective adjustment for follow-up improvement of flow pattern.

6. CONCLUSION

1. Pumping station forebay has characteristic of curved flow motion. The main flow under the inertia effect of turning-toward moves to the lateral-outer which results in separation at the inner and leads to large scale recirculation which destroys the uniformity of velocity distribution in front of pumping station. It affects seriously the safe operation of pump units.

2. Adopting diversion pier and bottom sill combination measures can effectively change bend flow structure and eliminate large-scale recirculation, improve significantly the flow conditions to ensure the safety of pump units and highly efficient operation.

3. CFD numerical simulation is applied to calculate the turbulent flow movement of pumping station forebay. 3D flow fields of forebay of intake channel are predicted. Comparing the computational results with experimental results a good agreement is found between flow patterns obtained from calculation and test observation. This indicates that CFD numerical simulation for pumping station forebay is feasible.

Of course, apart from the flow pattern to verify the numeric simulation results, it is necessary to verify the particular velocity distribution of the numeric simulation for comparison. This requires flow measurements of two-dimensional or three-dimensional. It would be very difficult to carry out this task by using probe for single point measurement. The field instrument PIV can be a way to deal with this problem which needs further working. This will be helpful to the numerical simulation for improvement of accuracy and conformity with the actual situation.

ACKNOWLEDGMENTS

This project is supported by China Nature Science Funds under grant number 50779060 and JSUSF under grant number K9870.

REFERENCES

[1] Spurr K J W, F. Boucherat J, The application of vorticity concepts to three irrigation intake designs, proc. Inst. Civ. Engrs, 1976, Vol.61. 331-330.

[2] Li S, Lai Y, WeberL, et al. Validation of a 3D numerical model for water pump-intakes [J], Journal of Hydr. Research, 2004, 42 (2) : 2822-2921

[3] Ansar Matahel, Nakato Tatsuaki, Experimental study of 3D pump-intake flows with and without cross flow [J], Journal of Hydraulic Engineering, ASCE,2001, 127 (10) : 825-834

[4] Liu Chao, Cheng Li, Tang Fang ping. Numerical Simulation of Three-dimensional Turbulent Flow inside a Pumping Forebay[J]. Transactions of the ChineseSociety of Agricultral Machinery, 2001, 32(6):41-44.(in Chinese)

[5] Rajendran V P, Constantinescu G, Patel V C. Experimental validation of numerical model of flow in pump-intake bays [J], Journal of Hydraulic Engineering, ASCE, 1999, 125 (11) : 1119-1125

[6] Constantinescu G, Patel V C. Numerical model for simulation of pump-intake flow and vortices[J], Journal of Hydraulic Engineering, ASCE, 1998,124 (2) : 123-134

[7] Yu Yonghai, Xu Hui, Cheng Yongguang. CFD numerical simulation on modification of flow pattern with flow deflector at forebay of pumping station[J]. Water Resources and Hydropower Engineering. 2006, 37(09):41-43.(in Chinese)

[8] Cheng Li, Liu Chao, Zhou Jiren, Tang Fang ping, Yuan Jiabo. Discussion on the Side-direction Flow of Pumping Station and Improving of the Flow Pattern. Drainage and Irrigation Machinery. 2001, 19(1):31-34. (in Chinese)

[9] Hydrodynamic Engineering Laboratory, Research Report on Model Test of the Intake Channel Flow of Jiangdu Pumping Station, 2006.12.

[10] Zhou Longcai , Liu Shihe , Qiu Chuanxin Numerical Simulation of Flow Patten in the Front Inflow Forebay of Pumping Station. Drainage and Irrigation Machinery. 2004, 22(1):23-27. (in Chinese)

