Three-dimensional numerical simulation of flow in continuous bend under the influence of piers

Sai Hua Huang1,3, Wen Wei Chai1,2,3, and Jun He2*

1College of Hydraulic and Environmental Engineering, Zhejiang University of Water Resources and Electric Power, Hangzhou 310018, China
2College of Hydraulic and Environmental Engineering, Three Gorges University, Yichang 443002, China
3Key Laboratory for Technology in Rural Water Management of Zhejiang Province, Hangzhou 310018, China
E-mail: hejun50@163.com

Abstract. With the development of social economy, more and more cities plan to build various kinds of Bridges in curved rivers to construct urban traffic network. The Realizable $k$-$\varepsilon$ turbulence model and VOF method were used to capture the free water surface, Simple algorithm was used for coupling calculation to simulate the flow characteristics of continuous bend with piers placed at 45° forward bend section. Simulation results show that: (1) under the condition of continuous curve, set in the horizontal surface slope will change the channel distribution of piers, bridge piers and curve superposition, the piers downstream transverse surface slope significantly improved, but the bridge pier near area, on the other hand, because of the water resistance of piers, bridge pier influence and affect the part of the balance of the bend, transverse surface slope value is only under the condition of no pier 1/2;(2) The high longitudinal flow velocity area is more concentrated, the high velocity center moves down to the bottom of the channel, and the central position of the circulation structure moves to the convex bank, which can aggravate the scouring of the convex bank.

1. Introduction

Almost all rivers in nature are curved, and bends can be regarded as the most basic unit of rivers [1]. Due to the special boundary conditions, the flow structure of curved channel presents irregular three-dimensional characteristics, and the water and sediment transport law and the evolution characteristics of riverbed scouring and silting are complicated and changeable. For this reason, many scholars have studied the flow characteristics of bend. Tan Z L et al. used the $k$-$\omega$ vorticity-viscous-turbulent model combined with the VOF method to simulate the three-dimensional characteristics of the water flow in the 180° bend of the open channel, and effectively predicted the variation range of the water surface height and the main influence range of the horizontal drop of water surface [2]. Wei Wen li et al. used the large eddy model of gas-liquid two-phase flow to simulate the 90° open channel in three dimensions, and accurately simulated the free water surface of the curve by VOF method [3]. Qin C C [4] et al. applied the modified model to the flow simulation of the continuous bend with large curvature and similar to the plane shape of natural river channel, and the model has a strong applicability to the simulation and analysis of the transverse circulation structure of the bend. Yang F [5] et al. used the semi-Lagrangian method based on Gauss point to calculate the...
convection term, and simplified the three-dimensional model on the premise of ensuring the calculation accuracy, and the calculation efficiency was significantly improved. Zhang B C et al. simulated the sinusoidal derived curve curve with large aspect ratio by using LES model, and effectively predicted the mainstream core area of the curve with large aspect ratio. According to the curve model based on previous studies, water and sediment movement characteristics existing in corner although there are a lot of research results, but basically is aimed at a single simple curves is given priority to, and with the development of social economy, due to various reasons, bridge construction is often difficult to avoid the curved reach, this makes the simple curves of bend flow characteristics has significant limitations, so the bridge pier under the influence of continuous flow structure and sediment transport law remains to be further research. In order to have a deeper understanding of the influence of bridge piers on the flow law of continuous bend, this paper conducts three-dimensional numerical simulation after placing the bridge piers at the 45° forward bend section of continuous bend.

2. Mathematical model and calculation method
The incompressible continuity equation, the motion equation and the equation that can realize the k-ε equation model are described as follows:

Continuity equation:

$$\frac{\partial u_i}{\partial x_i} = 0$$  \hspace{1cm} (1)

Motion equation:

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( (v + \nu_t) \frac{\partial u_i}{\partial x_j} \right) + g_i$$  \hspace{1cm} (2)

In the formula, $u$ is the velocity of water flow in m/s, $\rho$ is the density of water in kg/m$^3$, $t$ is time in s, $x$ is the spatial coordinate; $v$ is the dynamic viscosity coefficient of water; $\nu_t$ is the turbulence viscosity coefficient of water; $g_i$ is the mass force component.

Considering the Realizable k-epsilon turbulence model in the introduction of more accurate formula of turbulent viscosity and the turbulent kinetic energy dissipation rate equation, and satisfy the constraint conditions of Reynolds stress, can be keep consistent with real turbulent Reynolds stress, more than the standard k-epsilon turbulence model is suitable for simulating vortex, separated flow, secondary flow and other complex flow phenomenon, this paper adopts the Realizable k-epsilon turbulence model. The turbulence model equation is as follows:

$$\rho \frac{dk}{dt} = \frac{\partial}{\partial x_j} \left[ (v + \nu_t) \frac{\partial k}{\partial x_j} \right] G_k + G_h - \rho \varepsilon - Y_M$$  \hspace{1cm} (3)

$$\rho \frac{d\varepsilon}{dt} = \frac{\partial}{\partial x_j} \left[ (v + \nu_t) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S_k - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\mu e}} + C_3 \frac{\varepsilon}{k} C_{\varepsilon} G_h$$  \hspace{1cm} (4)

In the formula, $G_k$ represents the turbulent kinetic energy generated by the average velocity gradient of the flow; $G_h$ represents the turbulent kinetic energy due to buoyancy and uplift; $Y_M$ represents the influence of compressible turbulent pulsating expansion on the overall dissipation rate; $C_1, C_2, C_3$ is a constant coefficient, and the constant values are 1.44, 1.92 and 0.09 respectively in calculation.

Transverse drop of water surface at bend is an important feature of flow characteristics. In order to study the transverse drop of water surface at bend, it is necessary to simulate the free surface of water flow, so it is necessary to track the free surface of water flow when studying the movement characteristics of water flow at bend. VOF model is a free surface tracking method based on fixed mesh, which simulates two or more non-mixable fluids by solving a separate momentum equation and
dealing with the volume fraction of each fluid passing through the region\([8]\). It is suitable for the simulation of layered or free surface flows, and its function satisfies the equation:

$$\frac{\partial \alpha_i}{\partial t} + u \frac{\partial \alpha_i}{\partial x} = 0$$  \hspace{1cm} (5)

3. Model establishment and verification

Based on the simulation of two continuous bends connected by 90°, this paper simulates the continuous bend with the influence of bridge piers. The width of the continuous bend is \(B = 30\ m\), and a transition straight section with a length of 200 m is set upstream of the forward bend entrance of the continuous bend, and the forward bend outlet and the backward bend entrance are connected. The cross-sections of the model channels are rectangular, the diameter of the pier is 2m. In order to analyze the influence of bridge piers on the water and sediment movement of the continuous bend, a circular bridge pier was set at the 45° forward bend section of the continuous bend. See Figure 1 for the specific model calculation scheme.

![Figure 1. Schematic diagram of continuous curve model. (a) No bridge pier, (b) With bridge pier](image)

The water flow in the VOF simulation of continuous bend is water-gas two-phase flow. The water inlet is set as the velocity inlet boundary condition, and the air inlet is set as the pressure inlet boundary condition. The water depth is \(h = 10.0\ m\), the water inlet velocity is constant distribution, and the magnitude is \(v = 0.62\ m/s\). The bottom of the continuous bend, the bridge pier and the two sides of the bend are all set as smooth wall surfaces without slip, and the viscous bottom layer is treated by standard surface function method. Air outlet and water outlet are set as pressure outlet conditions; The air inlet on the upper part of the continuous bend model is set as the pressure inlet, and the pressure is set as 1 standard atmosphere.

![Figure 2. Verification of average vertical velocity in Shukry bend. (a) Value of calculation, (b) Measured value.](image)

In this paper, the water-gas two-phase VOF model of continuous bend was validated by the classical Shukry 180° rectangular bend flume test\([9]\). The size of the Shukry rectangular bend test
model is: the width of the flume channel is 0.3 m, the height is 0.5 m, the inner diameter of the bend is 0.15 m, the outer diameter is 0.45 m, and the length of the straight line segment is 1.07 m. In the test, the water level at the upstream inlet of the model is 0.3 m, the inlet flow is 70 L/s, the corresponding average flow velocity at the inlet section is 0.778 m/s, and the control water level at the downstream outlet is 0.28 m. During calculation, the known velocity value is given at the inlet of the bend, the outflow boundary is set as the pressure outlet, and the standard wall function is adopted at the wall surface of the bend. Figure 2 for Shukry corners in longitudinal vertical mean velocity isoline distribution simulation was compared with the measured values, after comparing the two figures as you can see, the simulation of velocity distribution and measured values of Shukry bend test, the data base, so the model simulation of flow characteristics is reasonable and reliable.

4. Interpretation of result

4.1. Analysis of horizontal gradient of water surface

In order to study the influence of bridge piers on horizontal descent of water surface of continuous bend, water level maps of 0°, 45° and 90° sections of the first bend and 45° and 90° sections of the second bend were taken for comparative analysis under various working conditions.

Figure 3. Water level diagram of each typical section of continuous curve. (a) Forward bend 45° section, (b) Forward bend 90° section, (c) Backbend 45° section, (d) Backbend 90° section.

Figure 3 shows that, under the same initial conditions and velocity, the water level at the section before the bridge pier rises with the flow of water. Due to destruction of bridge pier on throughout the entire cross section of the circulation, so as to make the bridge pier near the horizontal surface slope area is reduced, the horizontal surface slope value is only under the condition of no pier 1/2, along
with the water flow to leave after a certain distance, piers can be seen in the figure, 45° bend section after continuous curves, two schemes of the horizontal surface slope has reached a considerable level, so that the water in the region of the outflow of bridge pier, transverse surface slope is a process of accelerated development.

4.2. Longitudinal velocity analysis

In order to study the change rule of the influence of bridge piers on the longitudinal velocity of continuous bend, the distribution map of the longitudinal velocity of water flow along the curve of each scheme is drawn.

As can be seen from Figure 4, the maximum longitudinal flow velocity in the continuous bend can reach 0.85 m/s after the construction of bridge piers, and the maximum longitudinal flow velocity in
the continuous bend increases by 0.05 m/s compared with the condition without bridge piers. Under the condition of no pier at the same time, the longitudinal velocity of flow is mainly concentrated in about 0.5 times the depth of the water, and after the construction of bridge pier, the longitudinal velocity of high-speed flow area is also becoming more concentrated, high velocity center moved to the bottom of the river close to 1.1 m, and maximum flow someone who comported himself, longitudinal velocity is also becoming more and more concentrated, water will continue to erode the bottom of the river.

4.3. Analysis of bend circulation mechanism
Due to the existence of the circulation of the bend, the curve of concave bank is washed by water flow, water scouring sediment transported to convex bank with flow of or with the mainstream sports to continuous curve under a convex bank, makes convex bank sediment deposition, therefore, the existence of the circulation of the bend, is the formation of concave bank scour, the root cause of the convex bank deposition[10]. The longitudinal velocity of the flow field is ignored in the analysis of the transverse circulation of the curve, and the flow field diagram of the transverse circulation of the continuous curve is obtained.

Figure 5. Velocity field diagram at each typical section of a continuous bend without piers. (a) Forward bend 45° section, (b) Forward bend 90° section, (c) Backbend 45° section.
Figure 6. Velocity field diagram of each typical section of continuous bend under the influence of piers (a) Forward bend 45° section, (b) Forward bend 90° section, (c) Backbend 45° section.

By comparing the velocity fields of each typical section of the continuous bend under the two conditions, it can be seen that, under the condition of no pier, the circulation structure of each section of the forward bend of the continuous bend presents that the upper water flows to the concave bank, while the lower water flows to the convex bank, forming a large circulation in the counter-clockwise direction. Throughout the cross section after the construction of bridge pier in continuous curve, the circulation is segmented into three directions of the same small circulation, as the water flow movement, and the continuous curves before 90° section, can be seen from the figure three small circulation again to a big circulation structure, circulation strength increasing, and after the construction of bridge pier, the former center of the curved circulation structure appeared to the convex bank migration trend. In the backbend area of a continuous bend, the strength and influence range of the counterclockwise circulation structure in the backbend are greater than that without piers, because the construction of piers increases the strength of the forward-bend circulation structure, and the strength of the clockwise circulation structure generated by the backbend itself is only 1/2 of that without piers.
5. Interpretation of result

In this paper, by means of numerical simulation method, the three-dimensional simulation research on the flow characteristics of continuous bend under the influence of bridge piers is carried out. The calculation mainly includes two parts: the three-dimensional simulation of the flow movement of continuous bend under the influence of bridge piers and the three-dimensional simulation of the flow movement of continuous bend under the influence of bridge piers. The simulation results under the two conditions were compared and analyzed, and the following conclusions were drawn:

1) Bridge piers have a great influence on the transverse drop of water surface in the continuous bend. The transverse drop of water surface in the area near the piers decreases to only 1/2 of that without piers. After the water leaves the piers, there is a process of accelerated development of the transverse drop of water surface.

2) Under the condition of no bridge pier, the longitudinal velocity of flow is mainly concentrated in about 0.5 times the depth of the water, and after the construction of bridge pier, the longitudinal velocity of high-speed flow area is becoming more concentrated, the largest longitudinal flow velocity increased by 0.5 m/s, high velocity center moved to the bottom of the river nearly 1.1 m, so the water will continue to erode the river at the bottom of the convex bank.

3) After the bridge piers were built on the continuous bend, the circulation through the whole section was divided into three small circulations, which developed into a large circulation structure with the movement of water flow, and the center of the forward bend circulation structure showed a tendency to shift toward the convex bank. In the backbend area of the continuous bend, the development of the circulation structure of the backbend itself lags behind that of the non-pier condition.

References

[1] Tu Y and Liu C J. 2021. On the Distribution of Surface Flow Velocity in 180° Curved Open Channe. Science Technology and Engineering. 21(4) 1543-47.

[2] Tan Z L and Peng Y. 2012. Three-dimensional numerical simulation of bend flow in open channel. Port & Waterway Engineering. 3 46-49.

[3] Wei W L, Bai C W, Zheng Y, Liu Y L and Rong G W. 2015. Simulation of transverse circulation of 90° open channel using two phase gas-liquid large eddy model. Chinese Journal Of Applied Mechanics. 32(3) 466-472+10.

[4] Qin C C, Shao X J, Zhou J Y, Zhou G and Xiao Y. 2016. Suitability of simulation flow in meandering channel by modified depth-averaged model. J Tsinghua Univ(Sci & Technol). 56(12) 1264-70+77.

[5] Yang F and Fu X D. 2018. 3-D hydrodynamic model using the spectral method in the vertical direction for bend flow simulations. J Tsinghua Univ(Sci & Technol). 58(10) 914-920.

[6] Zhou B C, Xu D, Ji C and Bai Y C. 2019. Large eddy simulations of hydrodynamic structure in channel bends with large width-depth ratios and variable curvatures. Journal of Hydroelectric Engineering. 38(6) 77-91.

[7] Wei W L, Guo Y Y, Zhang Z W, Liu Y L and Li D X. 2017. Hydraulic characteristics of straight and T-shape spur dikes in 60° channel bend by numerical simulation. Journal of Hydroelectric Engineering. 36(9) 91-99.

[8] Gao X P, Jing S G and Jia L F. 2014. Study on factors influencing the flow over bend spillway. Journal of Hydroelectric Engineering.33(4) 132-138.

[9] Shukry A. 1950. Flow around bends in an open flume. Trans Asce. 115(11) 751-779.

[10] Yu Y, Ai C F and Jin S.2018. Influence of Curvature on Curved Channel Flow Structure by 3D Numerical Simulation. Journal of Water Resources and Architectural Engineering. 16(3) 218-223.