Numerical study on the influence of anti-collision pontoon on the vortex behind pier tail and horse-shoe vortex around pier

Guang Yang¹, a, Linzhao Shu², b, Nan Ji³, c*, Zhipeng Qian⁴, d

¹School of Shipping and Naval Architecture, Chongqing Jiaotong University, Chongqing, China
²School of Shipping and Naval Architecture, Chongqing Jiaotong University, Chongqing, China
³School of Shipping and Naval Architecture, Chongqing Jiaotong University, Chongqing, China
⁴School of Shipping and Naval Architecture, Chongqing Jiaotong University, Chongqing, China

a815293283@qq.com
b478903658@qq.com
c*Corresponding author: 2554617008@qq.com
d872268573@qq.com

Abstract—In order to study the influence of pier anti-collision pontoon on the vortex behind pier tail and horse-shoe vortex around pier, taking circular pier as an example, Reynolds stress (RSM) turbulence model is used to numerically simulate the flow around pier with anti-collision pontoon. Firstly, the rationality of the numerical simulation method and the accuracy of the results are verified by comparing with the cylindrical flow model test. Then, the bare pier is equipped with an anti-collision pontoon to calculate the flow around the pier under different water depths. The results show that the installation of anti-collision pontoon will change the vortex structure of the flow around the pier; the vortex in the pier tail will fuse obviously due to the interference of the floating tank, resulting in the non-stationary characteristics. When the water depth becomes shallow, the position of horse-shoe vortex in front of the pier will gradually close to the pier, and the bed erosion caused by horse-shoe vortex on both sides of the pier will be more serious.

1. INTRODUCTION

Ships and floating objects on the surface of the water hit the bridge piers from time to time, therefore, most bridge piers will be equipped with floating anti-collision boxes at the free surface in inland waterways. When the bridge piers are installed with anti-collision floating box, it changes the incoming flow near the free surface of the bridge piers, which will certainly have some influence on the bridge piers around the flow. Therefore, it is of great theoretical significance and engineering application value to study the influence of the anti-collision floating box on the structure of the bridge pier bypass tail vortex and horse-shoe vortex.
The problem of bridge pier bypass is essentially related to the problem of cylindrical bypass because they are the same in the flow nature. Yang Wanli et al [1] studied the characteristics of the resistance and lift of bridge pier column, and elaborated the relationship between the three-dimensional characteristics of the flow field and the pier column water flow force, and found that the pier column water flow force is non-uniformly distributed along the water depth. Kahraman et al [2] explored the effect of the free liquid surface on the flow field with the pointed pier as the object of study, and the results showed that the change of the free liquid surface height near the pier has a significant effect on the flow field in front of the pier and the wake region. Guemou et al [3] selected the DES model to simulate the bridge pier bypass flow numerically and found that the length of the bridge pier did not have any effect on the shear stress in the riverbed. Alemi and Maia [4] used the numerically stabilized large-step SSIIM program to study the bypass flow of single and complex piers and obtained more realistic results.

Some current studies are aimed at the characteristics of flow around bare piers, but the impact of pier equipped with anti-collision pontoon has not been deeply studied. Based on the computational fluid dynamics software star-ccm+, this paper simulates and calculates the flow around the pier with anti-collision pontoon in different water depths.

2. Mathematical Models and Numerical Methods

2.1 Mathematical Model

The numerical simulation uses the RANS equation for incompressible flow, three-dimensional non-constant with free liquid surface flow, and the controlling equations are the continuity equation and the Reynolds mean equation, as follows:

\[
\frac{\partial U_i}{\partial x_i} = 0
\]

(1)

\[
\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \mu \frac{\partial^2 U_i}{\partial x_j \partial x_j} + \frac{1}{\rho} \frac{\partial (\rho u_i u_j)}{\partial x_j}
\]

(2)

Where \( U_i \) is the mean velocity component, \( x_i \) and \( x_j \) is the coordinate component, \( \rho \) is the fluid density, \( p \) is the mean fluid pressure, \( \mu \) is the dynamic viscosity of the fluid, \( t \) is the time, and \( \rho \mu u_i u_j \) is the Reynolds stress term.

2.2 Turbulence Model

Chen et al [5] found that different turbulence models have a large impact on the calculation results of cylindrical winding flow, and the RSM model can better simulate the three-dimensional complex flow field around an upright cylinder compared with several commonly used turbulence models. The RSM model has a higher prediction accuracy for complex flows, discarding the assumption of isotropic vortex viscosity and closing the N-S by attaching the Reynolds stress transport equation and the dissipation rate equation with the following control equations:

\[
\frac{\partial}{\partial t} \left( \rho u_i \right) + \frac{\partial}{\partial x_k} \left( \rho u_i u_k \right) = D_{ij} + P_{ij} + \Pi_{ij} - \varepsilon_{ij}
\]

(3)

Where \( u_i \) and \( u_j \) are the velocities in the direction of \( i \) and \( j \) respectively, \( D_{ij} \) are the diffusion terms, \( P_{ij} \) are the generation terms, \( \Pi_{ij} \) are the pressure-strain terms, and \( \varepsilon_{ij} \) are the dissipation terms. Their expressions are as follows:

\[
D_{ij} = \frac{\partial}{\partial x_k} \left( \frac{\sigma_{ik} \sigma_{kj}}{\sigma_k} \right)
\]

(4)
\[ P_{ij} = -\left( u_{i} \frac{\partial u_{j}}{\partial x_{k}} + u_{j} \frac{\partial u_{i}}{\partial x_{k}} \right) \]  

\[ \Pi_{ij} = C_{1}e_{ij} + C_{2}e_{ij} + 1/3 \alpha_{ik} \alpha_{jk} \delta_{ij} \]  

\[ + C_{3}k_{ij} + C_{4}k_{ij} + 2/3 \alpha_{ik} \delta_{ij} \]  

\[ = \frac{2}{3} \rho \varepsilon_{ij} \]  

Where \( \alpha_{ij}, \alpha_{ik} \) and \( \alpha_{jk} \) are Reynolds stress anisotropy tensor, \( \delta_{ij} \) are turbulent stresses, \( s_{ij}, s_{ik}, s_{jk} \) are strain rate tensor, \( w_{ik} \) and \( w_{jk} \) are angular velocity tensor.

3. COMPUTATIONAL MODELING AND VALIDATION

3.1 Numerical Computational Modeling and Meshing

The computational model is shown in Figure 1. The origin of the computational domain is the position of the center of the cross-section of the cylinder on the free surface, the direction of \( x \) axis is the flow direction, the direction of \( y \) axis is the lateral direction, the direction of \( z \) axis is the vertical direction, and the diameter of the cylinder is \( D = 0.15 \) m. The length of the computational domain is \( 35D \), the upstream inlet of the cylinder is \( 10D \) from the center of the cylinder, and the downstream outlet is \( 25D \) from the center of the cylinder in the same direction as the flow direction ( \( x \) axis direction). The width of the computational domain is \( 10D \), and the distance between the left and right sides from the center of the cylinder is \( 5D \), and the direction is perpendicular to the direction of flow ( \( y \) axis direction). The water depth \( H = 0.20 \) m, the height of air above the free liquid surface is \( 0.10 \) m, the average flow velocity \( U_{0} = 0.26 \) m/s, the Reynolds number is \( 39000 \). The boundary inlet and the top are set as velocity inlet, the outlet is set as pressure outlet, and both sides of the computational domain, the column surface and the bottom surface are set as non-slip wall surface.

![Figure 1. Schematic diagram of flow around a cylinder.](image)

### TABLE I. GRID BASIC PARAMETER INFORMATION

| Mesh Quality | Total number of meshes | Number of internal surfaces | Number of vertical stratification |
|--------------|------------------------|-----------------------------|----------------------------------|
| rough        | 2 564 052              | 7 657 604                   | 30                               |
| medium       | 3 361 099              | 10 042 959                  | 32                               |
Figure 2. Mesh details around the cylinder.

The grid is adopted as a cut-body grid, and the grid is encrypted around the cylinder, the free liquid surface and the winding turbulent region, as shown in Figure 2, and the grid is gradually sparse in the region far from the cylinder, and the thickness of the boundary layer is taken as 0.067\(D\). In order to investigate the influence of the grid quantity on the calculation accuracy of the model, three kinds of grids with different qualities of coarse, medium and fine are obtained by changing the basic size of the grid, and the specific parameters of these three grids are given in Table 1.

### Table 1

| Mesh Quality | Total number of meshes | Number of internal surfaces | Number of vertical stratification |
|--------------|------------------------|-----------------------------|----------------------------------|
| fine         | 4764385                | 1424447                     | 35                               |

3.2 Model Validation

![Graph](image-url)
Figure 3. Transverse distribution of time averaged velocity at different positions downstream of a cylinder.

The model validation was performed using the cylindrical flow around experiment of Dargahi[6], and a plot of the time-averaged velocity transverse distribution at different locations downstream of the cylinder compared to the experimental values is given in Figure 3. The time-averaged flow velocity values are dimensionless by dividing by the transverse maximum flow velocity, where (a), (b), and (c) indicate the velocity distributions at different locations downstream of the cylinder near the free surface, and (d), (e), (f) indicate the velocity distributions at the corresponding locations near the bed surface.
From the figure, it can be found that the simulation results of the three meshes are in good agreement with the experimental values, and only the velocity error at the position directly behind the cylinder in Figure 3(f) is larger, which may be due to the difference between the roughness setting at the near-bed surface and the experiment. After considering the calculation accuracy and time cost, the medium grid property setting parameter is chosen as the grid division parameter for the subsequent working condition calculation.

3.3 Simulation Condition Setting and Calculation
The diameter of piers in bridge engineering is generally between 1m and 10m, the flow velocity is approximately between 1m/s and 3m/s, and the average water depth in each section of the Yangtze River channel is generally around 2m to 10m. In order to correspond to the validation conditions, selects 7.5m diameter piers as the object of study, and the scaling ratio is 1:50, then the diameter of the bridge pier of the calculation model \( D = 0.15m \), and the calculated water depth is between 0.04m and 0.2m. According to Fourier's law of similarity, when calculating the model flow velocity \( U = 0.26m/s \), corresponding to the actual flow velocity of 1.84m/s, the numerical model parameters have a good representation.

A circular cross-sectional anti-collision pontoon with a diameter of \( 2D \) is added to the bare pier, and the height of the floating box is set to 0.1m. In order to simplify the calculation, the draft of the floating pontoon is set to 0.05m, and it does not change with the change of water depth and other conditions. Three sets of water depth conditions are set for both the bare pier and the pier after adding the anti-collision pontoon, which are \( H = 0.20m, 0.15m \) and \( 0.10m \) respectively.

4. RESULTS AND DISCUSSION

4.1 Analysis of Three-Dimensional Vortex Structure behind Pier Tail
Figure 4 shows the 3D vortex cloud of the bare bridge pier at the same moment under different water depths. It can be seen from the figure that the bare bridge pier under the impact of water, the tail of three types of vortex, respectively, near the water surface and near the bed of the vortex behind pier tail, near the water surface of the side vortex, and near the bed of the horse-shoe vortex, and the smaller the water depth, the tail after the vortex and the side vortex faster approach and fusion; side vortex by the free surface of the impact, and the water surface. Figure 5 shows the 3D vortex cloud of the bridge pier bypassing flow with the anti-collision pontoon at the water depth \( H = 0.20m \). Due to the interference of the anti-collision pontoon, the regular shedding of the vortices at the end of the bridge pier disappears, and the vortices at the end of the whole pier occur obvious fusion, forming several strips of vortices moving backward; three horse-shoe vortices are generated at the first part of the bridge pier and floating box, which are located at the free surface directly in front of the anti-collision pontoon, at the junction between the bottom of the floating box and the bridge pier, and near the bed surface.

In order to study the non-constant characteristics of the tail vortex shedding of the bridge pier, Figure 6 gives a comparison diagram of the development of the three-dimensional vortex structure in one cycle for the bare pier and the bridge pier with collision avoidance floating box at water depth \( H = 0.20m \). From the figure, it can be found that the tail vortex of the bare pier periodical shedding is obvious, T/4 moment, after the pier tail near the wall vortex and boundary layer began to separate; T/2 moment, after the tail vortex began to gradually grow larger; 3T/4 moment, the vortex began to produce swing off; T moment, the shedding of the tail vortex backward extension development, new vortex continues to separate generated, and then began the next cycle. After the installation of anti-collision pontoon, the tail vortex of the bridge pier is disturbed, the vortex occurs obvious fusion, forming a band vortex backward movement development, the tail flow field due to interference by the floating box and lead to non-constant characteristics of the performance is not obvious.
Figure 4. Three-dimensional vorticity magnitude contours of flow field around bare piers.

Figure 5. Three-dimensional vorticity magnitude contours of flow field around piers with anti-collision pontoon.
4.2 Analysis of Local Horse-shoe Vortex of Bridge Pier

Figure 7(a) and (b) shows the Q-criterion clouds in the longitudinal section in front of the bare piers and the piers with collision avoidance floating box, and the vortex clouds in the longitudinal section in front of the piers can reflect the location and intensity of the horse-shoe vortex on the bed. The location of the horse-shoe vortex of the bare pier is between $0.13D$ and $0.31D$ in front of the pier, and the location of the horse-shoe vortex is between $0.11D$ and $0.27D$ in front of the pier after the addition of the crashworthy floating box, and there is no significant difference between the strength and range of the two. It is also known from the literature\cite{7} that when the anti-collision pontoon is added near the free surface, it has no significant effect on the horse-shoe vortex in front of the pier.
Figure 7(c) and (d) shows the Q criterion clouds for the pier after the installation of the anti-collision pontoon at water depths $H = 0.15m$ and $H = 0.10m$, respectively. The location of the horse-shoe vortex is between $0.08D$ and $0.23D$ in front of the pier at water depth $H = 0.15m$, and between $0.03D$ and $0.15D$ in front of the pier at water depth $H = 0.10m$, indicating that the horse-shoe vortex in front of the pier is gradually approaching the pier as the water depth decreases, and the intensity increases, and the scour pits on the bed will be closer to the pier and the scour is more serious.

5. CONCLUSIONS
We numerically simulate the installation of anti-collision pontoon on the pier and its flow around the pier in different water depths, and analyze the influence of anti-collision pontoon on the three-dimensional vortex structure and local horse-shoe vortex behind the pier. The main conclusions are as follows:

(a) The anti-collision pontoon will change the vortex structure around the flow tail of the pier. After the installation of the anti-collision pontoon, the vortex at the tail of the pier will merge obviously, forming a strip vortex band moving backward, and three horseshoe vortices will be generated at the free surface in front of the anti-collision pontoon, at the junction between the bottom of the pontoon and the pier and near the bed.

(b) After the installation of anti-collision pontoon, the vortex at the tail of the pier is disturbed, the vortex is obviously fused, and the unsteady characteristics of the tail flow field are not obvious due to the interference of the pontoon.

(c) The anti-collision pontoon has no obvious influence on the local horseshoe vortex of the pier, but the water depth has a great influence on the horse-shoe vortex. When the water depth becomes shallow, the position of horse-shoe vortex in front of the pier will gradually close to the pier.

ACKNOWLEDGMENT
This paper is supported by the graduate scientific research innovation project of Chongqing Jiaotong University in 2021 (2021S0044), and the Institute of Ship and Ocean Engineering Hydrodynamics of Chongqing Jiaotong University. Thanks for their support in my paper.

REFERENCES
[1] Yang Wanli, Wu Chengwei, Zhu Quanlong, Wang Guangjun. (2020) Refined study on 3D flow characteristics around bridge piers. Journal of Southwest Jiaotong University, 55(1): 134-143.
[2] R. Kahraman, M. Riella, G R. Tabor, M. Ebrahimi and P. Kripakaran. (2019) Prediction of flow around a sharp-nosed bridge pier: influence of the Froude number and free-surface variation on the flow field. Journal of Hydraulic Research, 2: 1–12.
[3] B. Guemou, A. Ghenim, A N. Ghenim. (2016) Numerical investigations of the round-nosed bridge pier length effects on the bed shear stress. Progress in Computational Fluid Dynamics An International Journal, 16(5): 313-321.
[4] A. Mahdi, M. Rodrigo. (2018) Numerical simulation of the flow and local scour process around single and complex bridge piers. International Journal of Civil Engineering, 16(5): 475-487.

[5] Chen Zhile, Liu Hua. (2008) Comparisons of different turbulence models based on the flow around a bottom-mounted cylinder. Chinese Journal of Hydrodynamics, 23(6): 633-638.

[6] B. Dargahi. (1989) The turbulent flow field around a circular cylinder. Experiments in Fluids, 8(1): 1-12.

[7] B. Dargahi. (1990) Controlling mechanism of local scouring. Journal of Hydraulic Engineering, 116(10): 1197-1214.